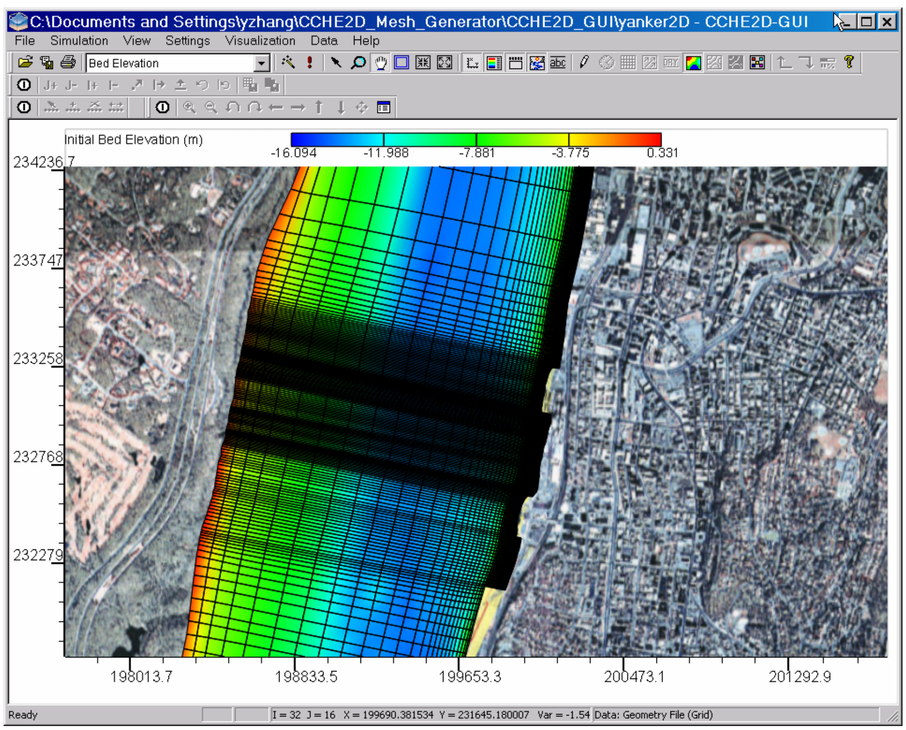
NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

CCHE2D-GUI – Graphical User Interface for the CCHE2D Model
User's Manual – Version 2.2

Technical Report No. NCCHE-TR-2005-03 May 2005

Yaoxin Zhang



School of Engineering
The University of Mississippi
University, MS 38677

NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

Technical Report No. NCCHE-TR-2005-03

CCHE2D-GUI – Graphical User Interface for the CCHE2D Model
User's Manual – Version 2.2

Yaoxin Zhang
Post-doctoral Research Associate

The University of Mississippi

May 2005

Table of Contents

1 Introduction

1.1 Family of CCHE2D Model

Welcome to the family of CCHE2D model, an integrated package for simulation and analysis of free surface flows, sediment transport and morphological processes. In addition to the numerical model itself, this family includes two more members: a mesh generator (CCHE2D Mesh Generator) and a Graphical Users Interface (CCHE2D-GUI), which will help you use CCHE2D model more easily and efficiently. The former, the CCHE2D mesh generator, allows the rapid creation of complex structured mesh systems for CCHE2D model with several integrated useful techniques and methods, while the latter, the CCHE2D-GUI, is a graphical users environment for the CCHE2D model with four main functions: preparation of initial conditions and boundary conditions, preparation of model parameters, run numerical simulations, and visualization of modeling results.

As illustrated in Fig. 1.1, the CCHE2D Mesh Generator provides meshes for CCHE2D-GUI and CCHE2D numerical model, while the CCHE2D-GUI provides a graphical interface to handle the data input and visualization for CCHE2D numerical model.

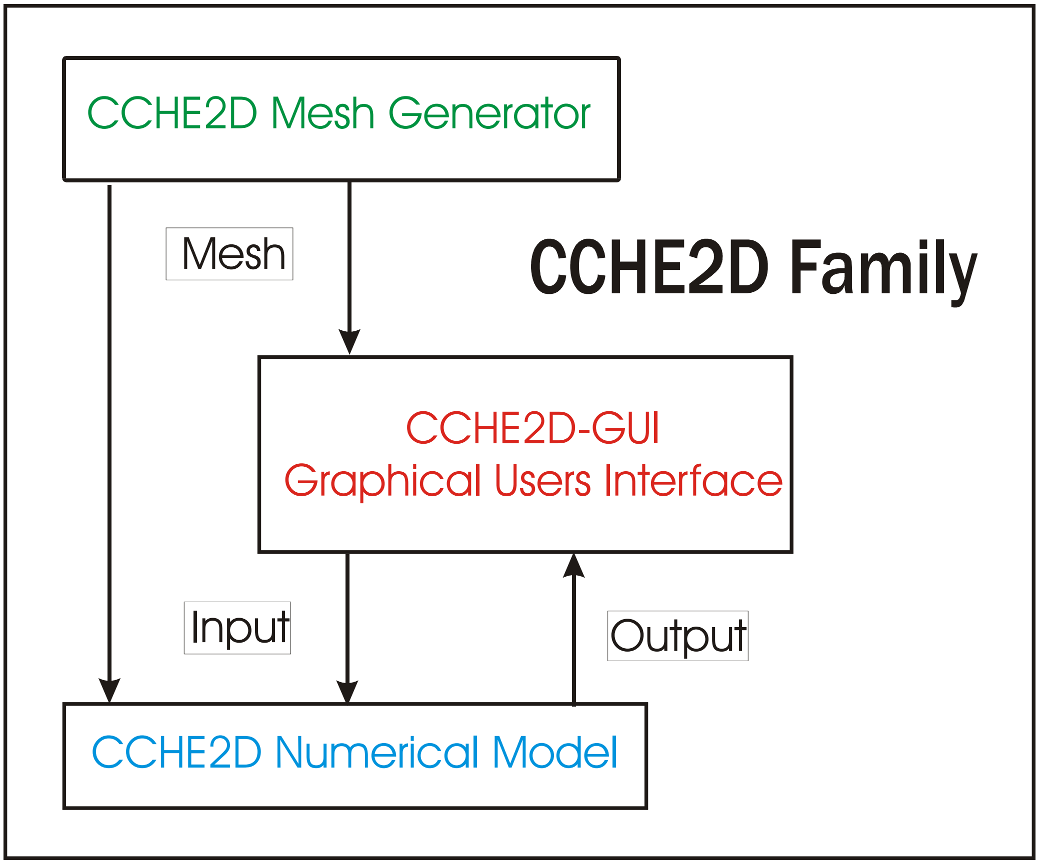


Figure 1

1.2 Using This Manual

This manual provides necessary information for using the CCHE2D-GUI, an intergrated 2D numerical modeling system. It explains in detail how to prepare and edit the initial conditions and the boundary conditions, how to set the model parameters, how to run CCHE2D model and how to visualize the modeling results.

This manual is organized as follows:

- Chapter 2 introduces the CCHE2D numerical model and the fundamentals on numerical simulation. Users new to numerical modeling are recommended to read this chapter carefully.
- Chapter 3 gives an overview of CCHE2D-GUI to make you familiar with the interactive graphical environment.

- Chapter 4 describes in detail the procedures to run the CCHE2D model through the GUI. You will learn how to edit grid, set initial conditions for flow, set model parameters, set the initial bed composition file and sediment boundary conditions files, set the initial bed material properties, set the monitor points, run simulations.
- Chapter 5 shows you how to visualize simulation results for flow and sediment transport, probe and extract data, set the flood properties and contour properties, edit the texts, set legend properties, and set the background color and background image.
- The appendix gives you a quick reference to the formats of files the users must provide for the CCHE2D-GUI.

Readers of this manual may also need to read the manual of CCHE2D Mesh Generator, a separate program that generates the numerical meshes required to start using the CCHE2D-GUI.

1.3 Related Documents

The documentation of CCHE2D is separate into several publications designed to fulfill the needs of different audiences. They are simply listed as follows:

- "CCHE2D-GUI Quick Start Guide" is intended for the first-time users.
- "CCHE2D-GUI – Graphical Users Interface for CCHE2D Model - User's Manual" describes in detail the capabilities and How-Tos of CCHE2D-GUI.
- "CCHE2D: Two-dimensional Hydrodynamic and Sediment Transport Model for Unsteady Open Channel Flows Over Loose Bed" describes in detail the basic mathematics, numeric, hydraulics and sediment transport approaches.
- "CCHE2D Sediment Transport Model" describes in detail the governing equations, boundary conditions, numerical methods and empirical formulas of the CCHE2D non-equilibrium transport model of non-uniform sediment.
- "CCHE2D Mesh Generator Users' Manual" is aimed at how to generate computational meshes for the CCHE2D model. It is associated with a separate software: CCHE2D Mesh Generator.

2 CCHE2D Model

2.1 Numerical Simulation

2.1.1 Introduction

Modeling Free Surface Flows and Sediment Transport with numerical models are much easier and more efficient than conducting field study in natural waters. One needs only a computer, a numerical model and the data. However, you may soon find out it takes more than these to have realistic and meaningful numerical solutions.

Numerical models are established based on the conservation laws and mathematics, it has to deal with many true physical and mathematical parameters. One has to understand all these parameters and make sure all the parameters prepared for the simulations are in correct range.

A numerical model is an approximation of the real world physical processes, even for very simple physical problems, the accuracy for the simulated quantities such as flow velocities and water surface elevation is limited. One should understand and expect errors due to mathematical approximation (Reynolds average, depth-average, and truncation errors, etc) and physical approximations (vertical flow acceleration is negligible, turbulent closure schemes, etc) involving in formulating a numerical model. Although numerical model verification and validation procedures could eliminate possible errors in the computation code due to mistakes, the errors due to approximations are inevitable.

When a model is applied to field study, one has to calibrate the model with field data. Because the resistance to the flow is represented by a roughness coefficient which varies with properties of sediment, bed form, channel geometry, and vegetation, etc., this information has to be characterized and fit to the model. Calibrating the model and identifying the mean or distribution of resistance to the flow is always necessary. Since the real world is very complicated, one normally would not have complete channel roughness information.

Numerical models approximate physical problems, it has however all the components to represent the physics to be simulated. Mesh or grid is used to represent channels and bathymetry; inflow and outflow are defined at inlet and outlet sections as boundary conditions. All the above and model parameters have to be defined before a simulation can start. To make setting up a simulation case efficiently, a mesh generator and a User Graphic Interface have been developed.

2.1.2 General Procedure of Using CCHE2D Model

The numerical modeling based on solving the Navier-Stokes equations is an initial-boundary value problem. Users must provide initial conditions and the boundary conditions. The general procedure of a numerical simulation can be simply listed as follows:

- Mesh generation
- Specification of boundary condition
- Parameters setting
- Simulation
- Results visualization and interpretation.

2.1.2.1 Mesh Generation

A mesh represents a computational domain and the way the governing equations are discretized. To have a successful simulation, one has to prepare the mesh carefully, so that the following concerns are taken into consideration:

- The interested zones has sufficient resolution;
- Transition between areas of different densities is smooth;
- Inlet(s) and outlet(s) should be sufficiently far away from the zones of interest;
- The mesh should be smooth and orthogonal as much as it allows.

Mesh generation particularly for practical problems takes a lot of time, however, the time shall be paid off if good quality is achieved. In many cases, the simulation code will run with a low quality mesh but the results maybe less reliable.

2.1.2.2 Boundary Conditions

Computations are conducted in a limited portion of free surface flows, boundary conditions are the driving mechanisms with which the flow in the simulated area behave. Therefore, one should set boundary conditions as close to true physics as possible. It is often the distributions of boundary conditions are unknown (distribution of velocity of discharge in a cross-section), in these cases, mean flow properties are specified and the inlet and outlet sections should be set distant from the interested zone.

2.1.2.3 Parameter Setting

Numerical simulation is to reproduce true physics by solving mathematic equations, therefore, many physical parameters and numerical parameters are needed. Some physical parameters have been provided in the Graphic User Interface as default which should be treated as guidance only. Many have to be provided by users for their particular applications.

Users must also provide the parameters that control the simulation processes. The sediment transport parameters (size distribution of bed materials and loads) often have higher level of uncertainty then those for the flow; bed roughness parameters are often identified through calibration which will match the energy slope of the numerical model to the studied physical subject.

2.1.2.4 Simulation

When the mesh is ready, boundary conditions have specified, one is ready to start a round of simulation. Since the CCHE2D is a model using a time marching scheme, one should also re-exam the initial condition. Because the initial condition for a water flow in natural condition is unknown, cares have to be taken to make sure the guessed initial condition is reasonable, particularly for unsteady problems.

Stability is a problem the modeler must aware. When the time step is too large for a particular problem, the simulation will not continue or it may produce totally unreasonable results. In these situations, one should reduce the time step used and retry, until the solutions becomes stable. Although an implicit scheme is used in the CCHE2D model, time step can not be set arbitrarily.

2.1.2.5 Results Interpretation

It is debatable how much one can trust the obtained numerical simulation results: “Is it reasonable?” The answer to this is yes and no.

Numerical model is developed under certain assumptions, the results should be reliable if the simulated flow satisfies these assumptions. However, the natural is always complicated, these assumptions could never be satisfied one hundred percent. If the results are questioned, we recommend validating them by comparing with physical model measurements or field data.

Once the model is validated with site specific data, it could be used to study the trend of the flow or sediment transport processes by varying concerned parameters

2.2 CCHE2D Model

CCHE2D model is a two-dimensional hydrodynamic and sediment transport model for unsteady open channel flows over loose bed. Here a brief introduction of the CCHE2D model is presented, its details can be found in Jia and Wang (1999 and 2001).

2.2.1 Governing Equations

The depth integrated two-dimensional equations are solved in CCHE2D model.

Continuity Equation:

$$\frac{\partial Z}{\partial t}+\frac{\partial (hu)}{\partial x}+\frac{\partial (hv)}{\partial y}=0 \tag{2.1}$$

Momentum Equations:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-g\frac{\partial Z}{\partial x}+\frac{1}{h}[\frac{\partial (h\tau_{xx})}{\partial x}+\frac{\partial (h\tau_{xy})}{\partial y}]-\frac{\tau_{bx}}{\rho h}+f_{Cor}v \tag{2.2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-g\frac{\partial Z}{\partial y}+\frac{1}{h}[\frac{\partial (h\tau_{yx})}{\partial x}+\frac{\partial (h\tau_{yy})}{\partial y}]-\frac{\tau_{by}}{\rho h}-f_{Cor}u \tag{2.3}$$

where u and v are the depth-integrated velocity components in the x and y directions respectively; g is the gravitational acceleration; Z is the water surface elevation; ρ is water density; h is the local water depth; f_{Cor} is the Coriolis parameter; $\tau_{xx}, \tau_{xy}, \tau_{yx}$ and τ_{yy} are the depth integrated Reynolds stresses; and τ_{bx} and τ_{by} are shear stresses on the bed surface.

2.2.2 Turbulence Closure

In Equations (2.2) and (2.3), the Reynolds stresses are approximated based on Boussinesq's assumption:

$$\tau_{xx} = 2\nu_t \frac{\partial u}{\partial x} \tag{2.4a}$$

$$\tau_{xy} = \tau_{yx} = \nu_t \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{2.4b}$$

$$\tau_{yy} = 2\nu_t \frac{\partial v}{\partial y} \tag{2.4c}$$

2.2.2.1 Eddy Viscosity Model

There are two zero-equation eddy viscosity models adopted in the CCHE2D model. The first one is the depth-integrated parabolic model, in which the eddy viscosity ν_t is calculated by the following formula:

$$\nu_t = \frac{A_{xy}}{6} \kappa U^* h \tag{2.5}$$

where A_{xy} is an adjustable coefficient of eddy viscosity, κ is the von Karman constant, and U_* is the shear velocity.

The second eddy viscosity model is the depth-integrated Mixing Length model. The eddy viscosity ν_t is calculated by the following equation.

$$\nu_t = \bar{l}^2 \sqrt{2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial \overline{U}}{\partial z}\right)^2} \tag{2.6a}$$

$$\bar{l} = \frac{1}{h}\int \kappa z \sqrt{\left(1 - \frac{z}{h}\right)} dz = \kappa h \int_0^1 \lambda \sqrt{1-\lambda}\, d\lambda \approx 0.267 \kappa h \tag{2.6b}$$

$$\frac{\partial \overline{U}}{\partial z} = C_m \frac{U^*}{\kappa h} \tag{2.6c}$$

where C_m is a coefficient with a value of 2.34375 so that Equation (2.6) will cover Equation (2.5) in the case of a uniform flow in which all the horizontal velocity gradients vanish.

2.2.2.2 Two-dimensional $k-\varepsilon$ Model

In this model, differential equations are introduced for the turbulent kinetic energy k and the rate of dissipation of turbulent energy ε, where $k = \frac{1}{2}\overline{u_i' u_i'}$ and $\varepsilon = \mu_t \overline{\frac{\partial u_i'}{\partial x_j}\frac{\partial u_i'}{\partial x_j}}$.

The depth-integrated governing equations for k and ε are:

$$\frac{\partial k}{\partial t} + u\frac{\partial k}{\partial x} + v\frac{\partial k}{\partial y} - \frac{\partial}{\partial x}[\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x}] - \frac{\partial}{\partial y}[\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial y}] = P - \varepsilon + P_{kV} \tag{2.7}$$

$$\frac{\partial \varepsilon}{\partial t} + u\frac{\partial \varepsilon}{\partial x} + v\frac{\partial \varepsilon}{\partial y} - \frac{\partial}{\partial x}[\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}] - \frac{\partial}{\partial y}[\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}] = c_{1\varepsilon}\frac{\varepsilon}{k}P - c_{2\varepsilon}\frac{\varepsilon^2}{k} + P_{\varepsilon V} \tag{2.8}$$

where

$$P = -\overline{u_i' u_j'} u_{i,j} = \nu_t [2(\frac{\partial u}{\partial x})^2 + 2(\frac{\partial v}{\partial y})^2 + (\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y})^2] \tag{2.9}$$

$$P_{kV} = C_k \frac{U_*^3}{h} \qquad P_{\varepsilon V} = C_\varepsilon \frac{U_*^4}{h^2} \tag{2.10}$$

$$U_* = \sqrt{c_f (u^2 + v^2)} \tag{2.11}$$

$$C_k = \frac{1}{\sqrt{c_f}} \qquad C_\varepsilon = 3.6 \frac{c_{2\varepsilon}}{c_f^{3/4}} \sqrt{c_\mu} \tag{2.12}$$

and c_f is the friction coefficient.

From the local values of k and ε, a local eddy viscosity can be evaluated as

$$\nu_t = \frac{c_\mu k^2}{\varepsilon} \tag{2.13}$$

In the above equations, the following values are used for the empirical constants: $c_\mu = 0.09$, $c_{1\varepsilon} = 1.45$, $C_{2\varepsilon} = 1.90$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$.

2.2.3 Sediment Transport

A brief introduction of CCHE2D sediment transport is presented here. For details, please refer to the technical report by Wu (2001).

2.2.3.1 Total Load

According to the conventional classification, moving sediment is divided into suspended load and bed load along the vertical direction. The bed load is the part of the sediment moving on or near the bed by rolling, saltating or sliding, while the suspended load moves in suspensions, which physically occupies the water column along the flow depth above the bed load layer. For a more general application, the ***total load*** sediment transport is simulated in CCHE2D model.

2.2.3.2 Non-equilibrium Transport

Since the suspended load transport occurs mostly at a non-equilibrium state, it is usually simulated by non-equilibrium transport models. Different from most existing sediment transport models that assume a local equilibrium of the bed load transport, the CCHE2D model implements a full non-equilibrium transport model for both bed load and suspended load. The depth-integrated convection-diffusion equation of the suspended load transport and the continuity equation of bed load are solved in CCHE2D model.

2.2.3.3 Bed Material Sorting

The bed material gradation usually varies along the vertical, so the bed material above the non-erodable layer is divided into several layers, as shown in Figure 2-1. The top layer is the mixing layer, and the second is the subsurface layer. The variation of bed material gradation in the mixing layer can be described by a partial differential equation, while in other layers under the mixing layer the bed material gradation can be determined by using the mass conservation law.

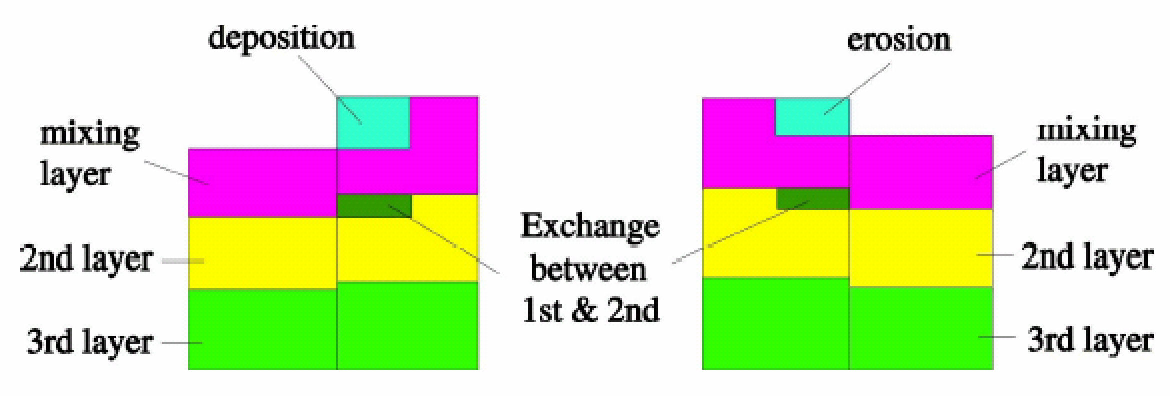


Figure 2-1

2.2.3.4 Initial Conditions

For a complete simulation of sediment transport, information on sediment properties, sediment transport capacity, non-equilibrium adaptation length and movable bed roughness should be given. The sediment properties include the sediment grain size, specific gravity (default value: 2.65), grain shape factor (default value: 0.7) and bed material porosity. The sediment transport capacity, non-equilibrium adaptation length and the movable roughness are determined by empirical formulas.

2.2.3.5 Empirical Formulas

Dozens of formulas for the fractional non-cohesive sediment transport are available. CCHE2D model selects four formulas or module capable of accounting for the hiding and exposure effect. The sediment transport capacity is determined by van Rijn's (1984) formula, Wu et al's (2000) formula, SEDTRA module (Garbrecht et al., 1995), the modified Ackers and White's formula (Proffit and Sutherland, 1983), or the modified Engelund and Hansen's formula (Wu and Vieira, 2000). The effect of secondary flow on the main flow and sediment transport in curved channels has also been considered in current version of CCHE2D model.

3 Overview of CCHE2D-GUI

3.1 Introduction

This chapter explores the interface of the CCHE2D-GUI. The functions of the main window, the toolbars, and the menus will be explained in details to help you get familiar with this interactive graphical environment.

3.2 Main Window

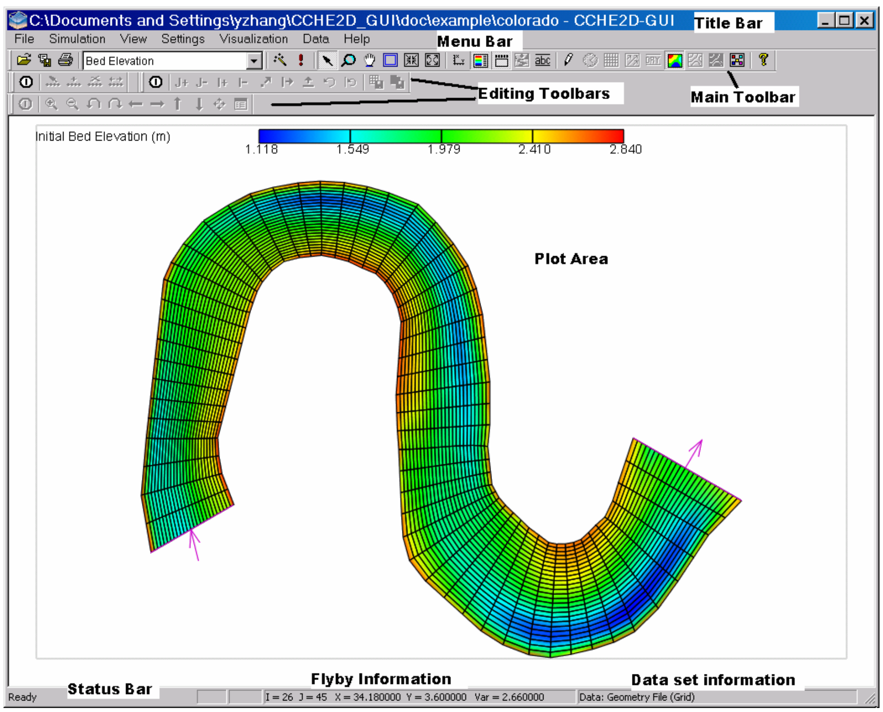
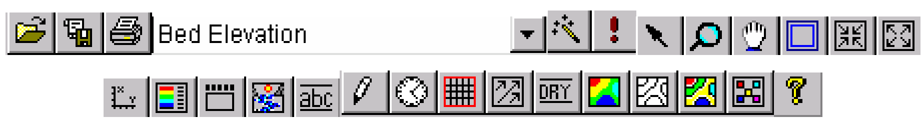


Figure 3-1

The main window provides an interactive platform for the users. As shown in Fig. 3-1, it contains all the functional components or the accesses to these components. It is composed of the title bar (displays the file path to the data set), the menu bar, the toolbars (main toolbar and editing toolbars), the status bar, and the plot area.

3.3 Toolbars

Main Toolbar



Bed Elevation	Display variable		Incremental Zoom Out		Plot Flood and Lines
	Open a Geo File		Add/Remove XY Axis		Plot Points
	Save as a Layout File		Add/Remove Legend		About
	Print Current Image		Add/Remove Title		
	Simulation Wizards		Show/Hide Background		Show/Hide Simulation Time
	Run simulations	abc	Add Texts	DRY	Show/Hide Dry Area
	Select Pointer		Data Probe		
	Zoom In		Grid On/Off		
	Pan		Vector On/Off		
	Zoom To Full Size		Plot Flood Contour		
	Incremental Zoom In		Plot Contour Line		

Toolbar for Grid Editing

	Start/Stop Grid Editing		Delete I Line		Undo Previous Change

	Add J Line		Move Grid Node		Undo All Changes
	Delete J Line		Move J Line		Save All Changes
	Add I Line		Move I Line		Save as a New Geo File

Toolbar for Boundary Editing					
	Start/Stop		Delete Boundary Node String		Modify Boundary Node String
	Add Boundary		Select Boundary Node String		

Toolbar for Background Image Editing					
	Start/Stop Editing		Rotate Right		Downward Translation
	Zoom In Image		Left Translation		Translation
	Zoom Out Image		Right Translation		Set Transformation Parameters
	Rotate Left		Upward Translation		

3.4 Menus

3.4.1 File

This is most probably the first menu you will touch when using CCHE2D-GUI. It provides almost all the accesses to files Input/Output, such as **opening** Geo and Layout files, **saving** Layout files, **importing** and **exporting** Bitmap file, **printing** current image, and etc.

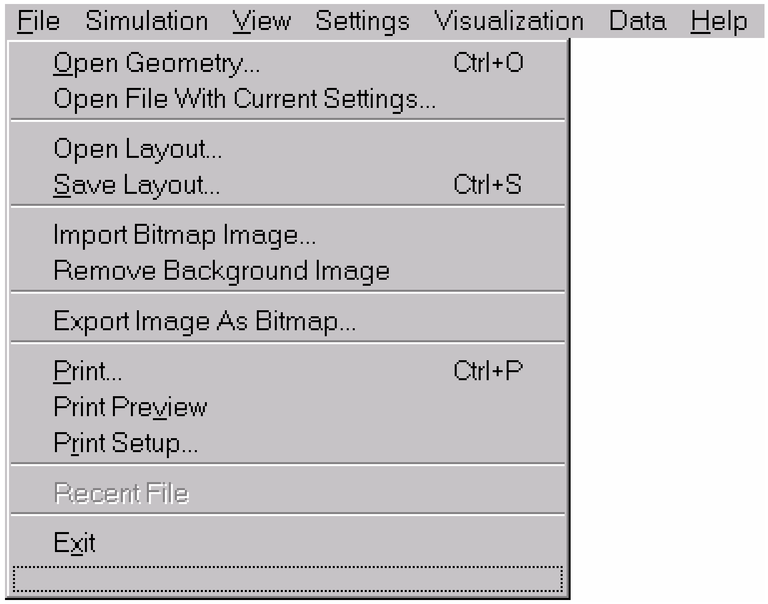


Figure 3-2

3.4.2 Simulation

After loading the Geo mesh file, you need to visit this menu to run simulations. According to the functions, the menu items are divided into four groups, “wizard” group, “run” group”, “flow” group and “sediment” group (see Figure 3-3).

In “wizard” group, it provides you a quick guide on how to run simulations.

In “run” group, it provides you access to run simulations, i.e., **Run CCHE2D Model…**.

In “flow” group, it provides you editing tools to prepare the initial conditions, boundary conditions and the model parameters for flow.

In “sediment” group, it provides you editing tools to prepare the initial conditions, boundary conditions and the model parameters for sediment.

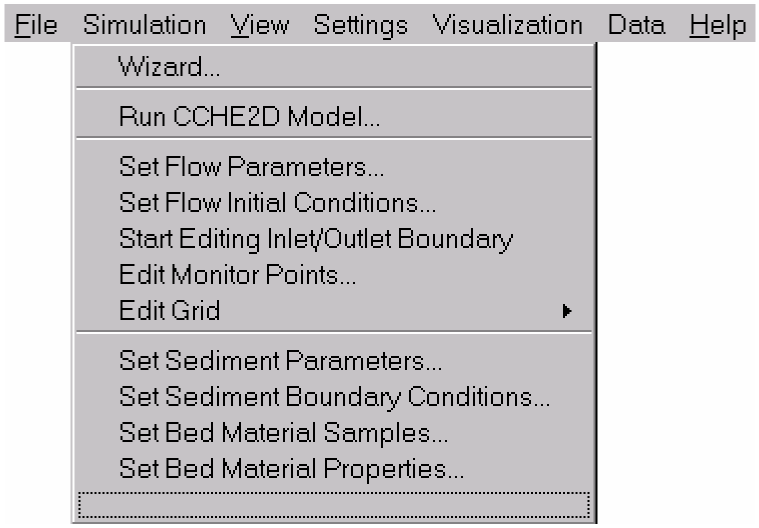


Figure 3-3

3.4.3 View

This menu provides you necessary tools to view the current image more clearly and accurately, such as **Select**, **Zoom**, and **Pan,** etc.

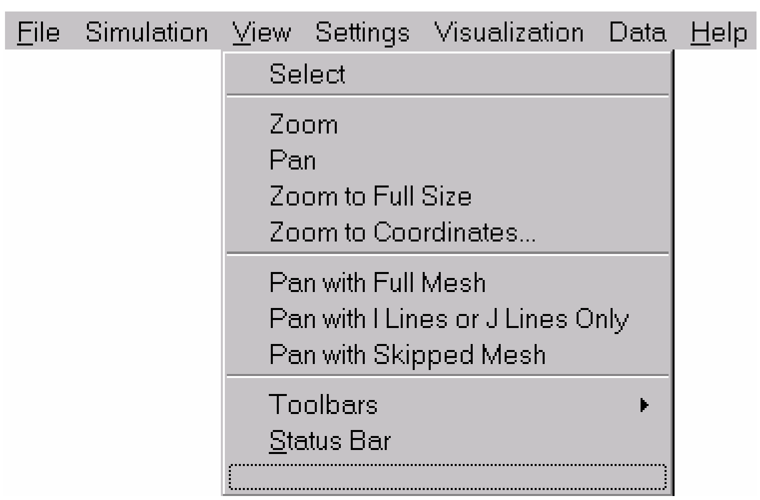


Figure 3-4

3.4.4 Settings

Visiting this menu, you can set up the visualization styles best fit your needs and personality, such as **plot type**, **colors**, adding **text**, showing/hiding **legend**, **title**, **grid**, **boundary, simulation time**, **vectors**, and etc. You can also set up all kinds of visualization parameters/properties, such as properties of **legend**, **title**, **text**, **flood**, **contour**, **vector**, **background image**, and etc.

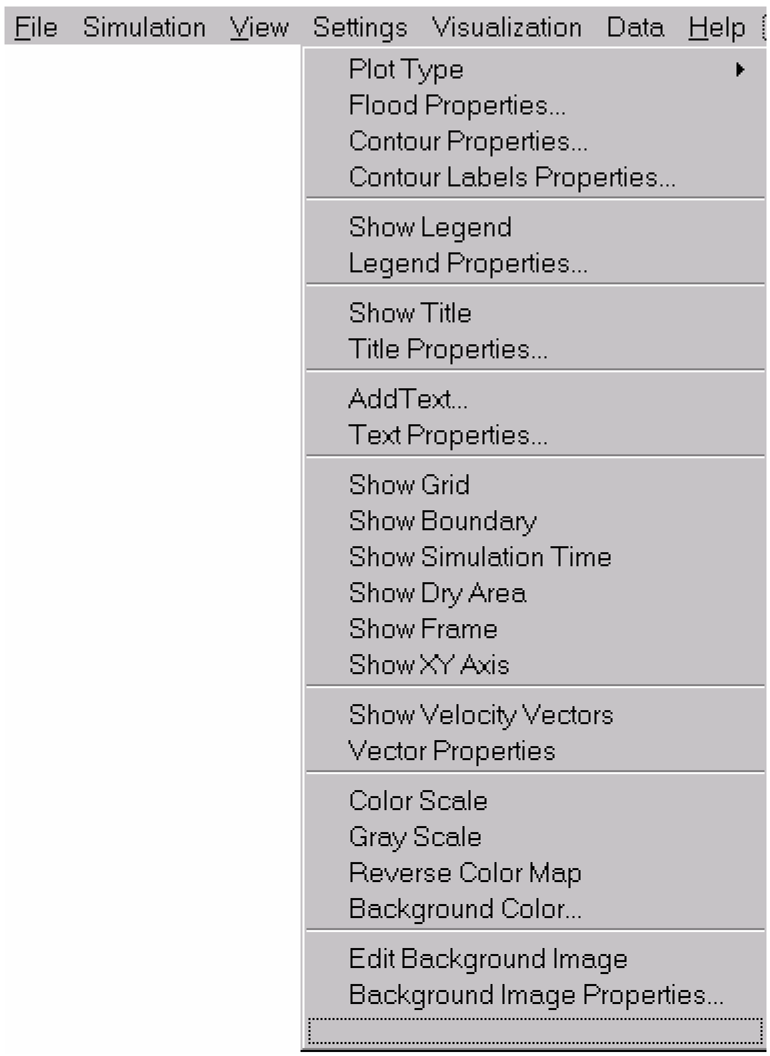


Figure 3-5

3.4.5 Visualization

Through this menu, you can visualize all kinds of available result files, the intermediate file, the final results file, and the history file.

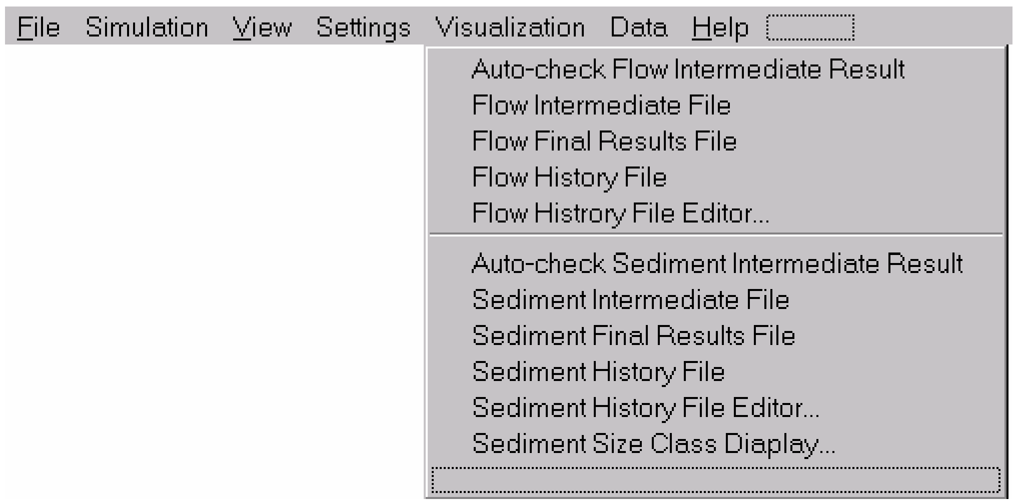


Figure 3-6

3.4.6 Data

You can **probe** and **extract** data from the image using this menu.

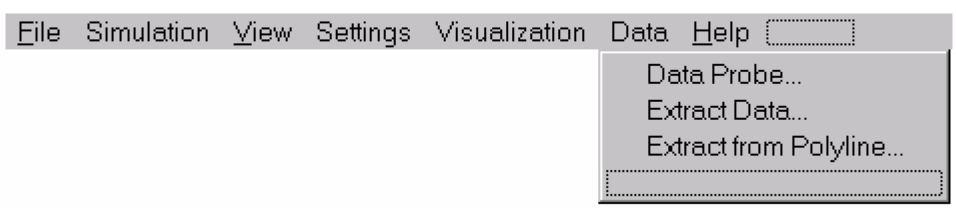


Figure 3-7

3.4.7 Help

Have any difficulties? Found any bugs? Any comments? **Contact us! Your suggestions and critics are very important to us**

3.4.8 Popup Menus

As shown in Figure 3-8, there are two popup menus: “toolbar” menu and “view” menu.

The “toolbar” popup menu is a copy of the submenu “**Toolbar”** in menu **“View”**, while the “View” popup menu is a simplified version of menu “**View**” for easy access. The former can be activated by right clicking the toolbar, while the latter will appear by right clicking the plot area.

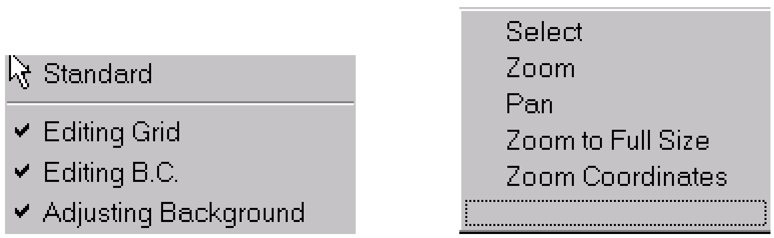


Figure 3-8

4 Run Simulations

4.1 Introduction

One main function of the CCHE2D-GUI is to run simulations using CCHE2D numerical model. Basically, there are five steps to run simulations. The first step in getting a case ready for simulation is to generate a mesh for the domain of interest. A mesh file, a file with a "**.geo**" extension, is required by both the CCHE2D-GUI and the CCHE2D numerical model. You need the help of the CCHE2D Mesh Generator. The mesh file contains the geometry and the initial conditions of the grid, such as the (x,y) coordinates, the initial bed elevation, the initial water surface level, the bed roughness and the nodal type. In this manual, **it is assumed that you already have such a mesh file.**

After loading the mesh file, you can **set the initial conditions** for flow, including editing the grid (refining, coarsening, and modifying nodal type), setting the initial bed elevation, the initial water surface level and the initial bed roughness.

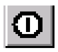
After generating and editing the mesh, the next step is to **set the model parameters.** The parameters are divided into two groups: flow parameters and sediment parameters. In flow parameters, you need to set the simulation parameters, the bed roughness and the general physical properties of water. In the sediment parameters, you need to define the sediment size classes, set the transport mode, the transport equations or formulas, and the sediment simulation options and parameters, etc.

Once the mesh generation process is complete and parameters for flow and sediment are set, you need to **specify proper initial and boundary conditions**. If sediment transport option is desired, you must specify initial composition of the bed. You can specify multiple inlet and outlet sections for the domain, prescribe specific discharge, total discharge, or discharge hydrograph at the inlet sections. At outlet sections, you can prescribe a constant water surface level, a open boundary condition, a rating curve, or a stage hydrograph. In addition, you can prescribe either steady or unsteady sediment boundary condition at the inlets.

After all the above steps are finished, the last step is to **run simulation**. You can run the CCHE2D model locally, that is on the same computer you are running the CCHE2D-GUI.

4.2 Edit Grid

The CCHE2D-GUI provides basic grid editing functionality that can be used for small modifications of the computational mesh. You should use the CCHE2D Mesh Generator to create the best possible mesh, and then import it into the CCHE2D-GUI.

After opening a mesh file, you can go to menu **Edit** and click **Start Editing** in submenu **Edit Grid** or click  to activate the toolbar for grid editing. The detailed explanations on each button of the toolbar can be found in chapter 3. You can make some fine tuning of the mesh before running simulations, such as adding or deleting mesh lines in either I or J direction, moving grid node, and moving I line and J line. If you want to regenerate the mesh, you need to use the CCHE2D Mesh Generator.

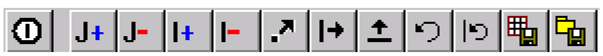
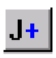
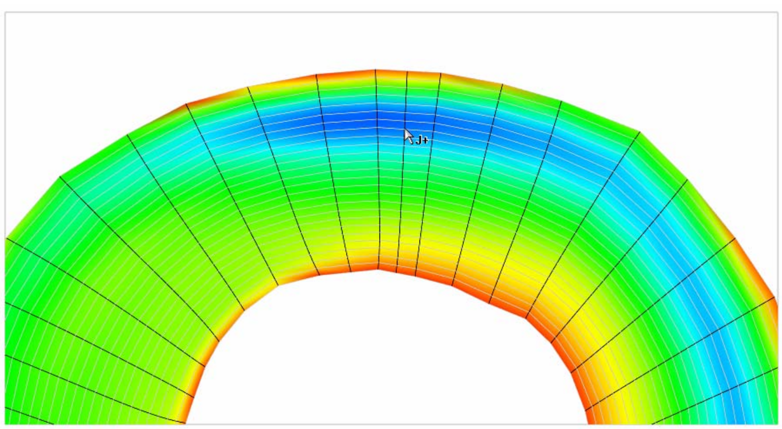


Figure 4-1

To **Add or Delete J Line**,

- Click  or  to enter the editing status. The I lines of the mesh will become gray while the J lines will become highlighted. The cursor will become  or .
- Click the desired place (actually somewhere in between two existing J lines) to add a J line.
- Click the J line you want to delete.



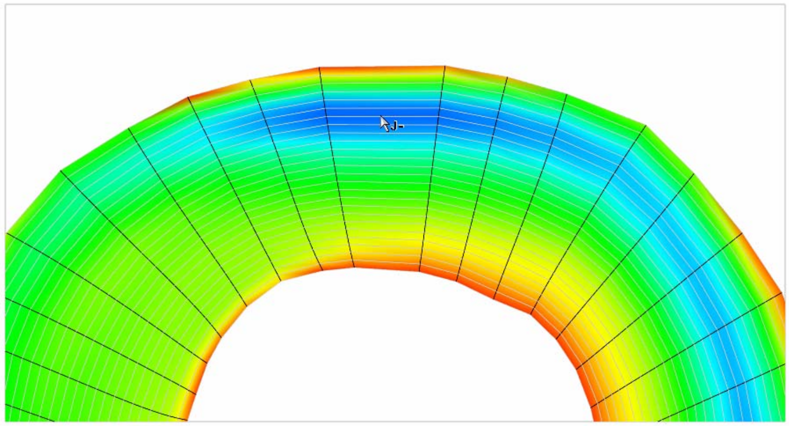
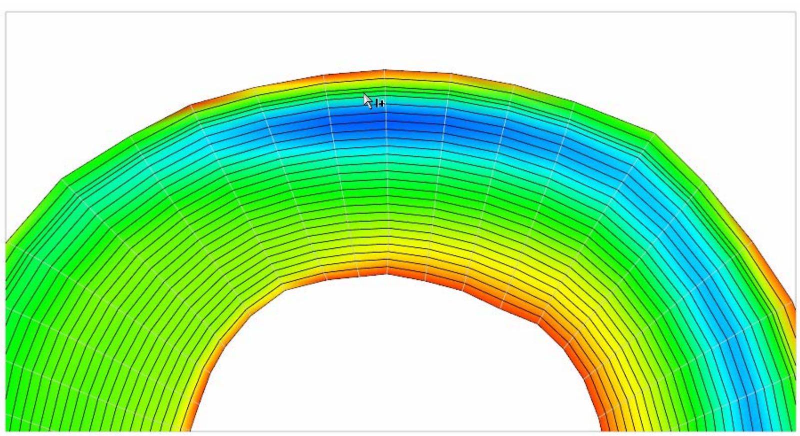


Figure 4-2

To **Add or Delete I Line**,

- Click or to enter the editing status. The J lines of the mesh will become gray while the I lines will become highlighted. The cursor will become or .
- Click the desired place (actually somewhere in between two existing I lines) to add an I line.
- Click a I line you want to delete.



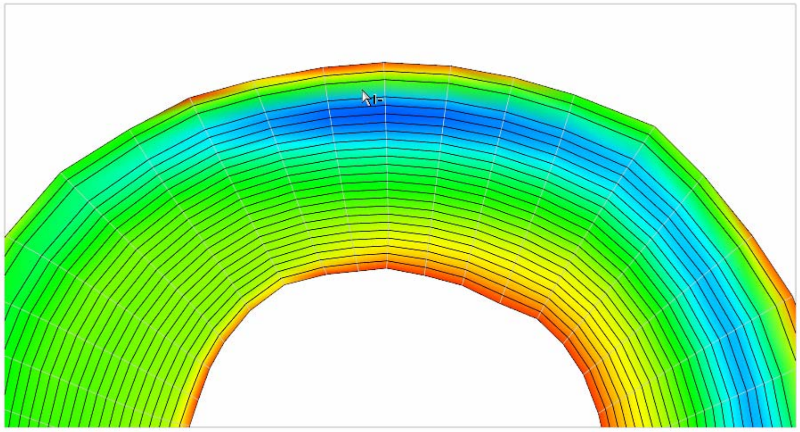
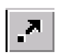
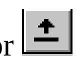
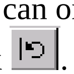
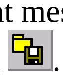


Figure 4-3

To **Move a Grid Node,** click  to enter editing status; then click the grid node you want to move and hold the mouse button; and drag it to the desired place and release the mouse button.

Moving a J line or I line is quite similar to moving grid node. First click  or  to enter editing status; then click the J line or I line you want to move and hold the mouse botton; and then drag it to the desired place and release the mouse button.

You can only **Undo one** previous step by clicking . If you want to **Undo All** the changes, click . You cannot undo the changes after you saved the changes. If you want to save the changes to the current mesh, click . Another alternative is to save all the changes to a new mesh file by clicking .

Note: adding or deleting mesh lines change the mesh size and the orderings of the mesh nodes. As a result, the boundary conditions and the results associated with the old mesh will be incompatible with the new mesh. And, moving mesh lines or nodes may degenerate the mesh quality and hence the accuracy of the solutions. Therefore, you must be very careful with the grid editing.

4.3 Set Flow Initial Conditions

The numerical simulations of flow and sediment transport based on solving Navier-Stokes Equations are initial-boundary value problems. The initial conditions and the boundary conditions must be specified before running the simulations. The initial conditions are very important, since inappropriate initial conditions may slow down the convergence process or even cause the simulation to fail. The initial conditions include the flow initial conditions and the sediment initial conditions. This section will tell you how to set the flow initial conditions.

In addition to the geometry information, the mesh file also contains the information of the initial flow conditions that include the initial water surface level, bed roughness, and initial bed elevation. To set the initial conditions for flow, you need to visit **Set Flow Initial Conditions…**in **Simulation** menu which activates the **Nodal Properties** dialog window.

To **Specify Initial Water Surface Level, Roughness and Bed Elevation,**

- Select the flow variable you want to specify by checking the radio button in **Assign WSL, Roughness or Bed Elevation.**
- If you click **Whole Domain**, the selected flow variable will be assigned a constant value through the domain.
- You can also give a spatial distribution to the selected flow variable by clicking **Sub-Region**. After clicking this button, the GUI waits for you to define the region by **selecting two points** which defines a rectangular area within the mesh.
- The **Assign Value** dialog will appear right after clicking **Whole Domain** or after you click the second point when defining **Sub-Region**. Enter the desired value and click **OK.**
- You can repeat the above steps until the initial conditions of all the flow variables are set.
- **Note: Please make sure the initial water level at the inlets is higher than bed.**
- **Tip: If the initial WSL is too low, there will be many dry nodes at the beginning of the simulation. This may cause the simulation fail. To avoid it, specify the initial WSL a little higher to reduce the number of dry nodes.**

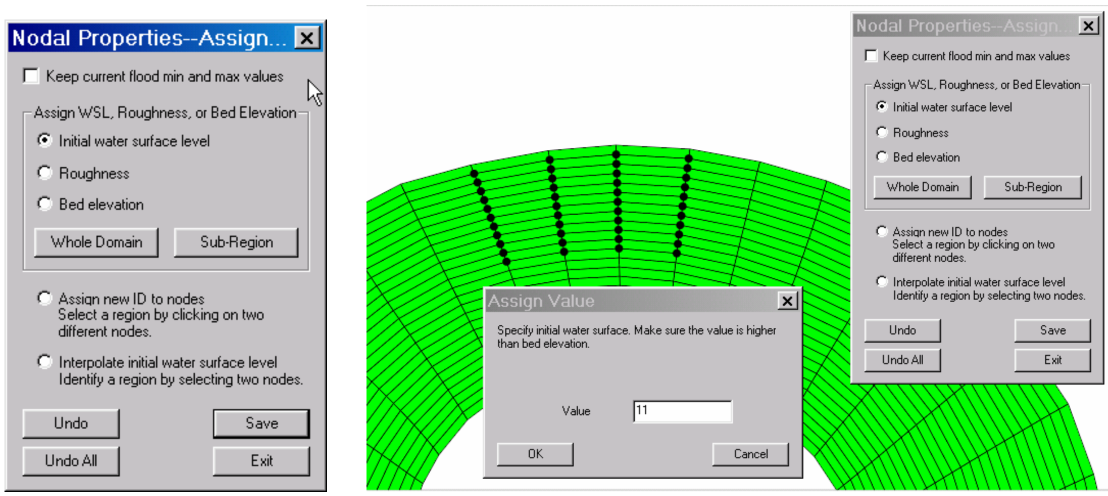


Figure 4-4

You can specify a **linear distribution** in either **I direction** or **J direction** of water surface level in a region represented by a pair of diagonal points instead of a constant distribution. To do the linear interpolation, check the option **Interpolate initial water surface level** and then click two different points to define a region. Finally choose the direction by checking **Interpolate along constant I-lines** or **Interpolation along constant J-lines** and click **OK.**

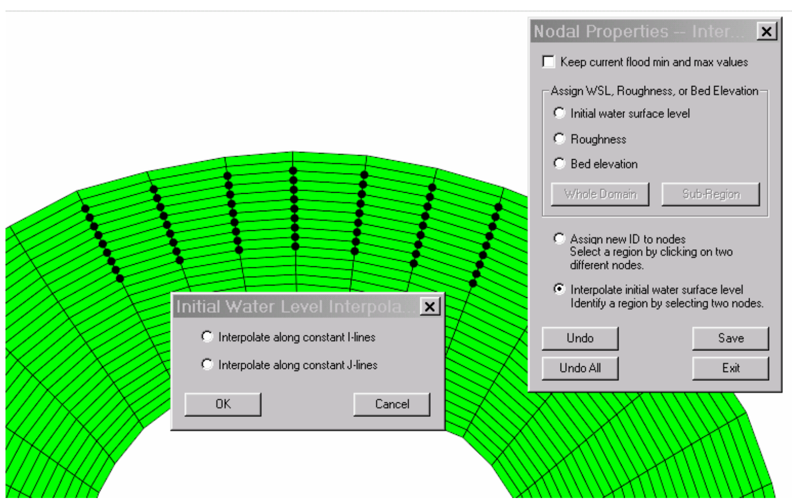
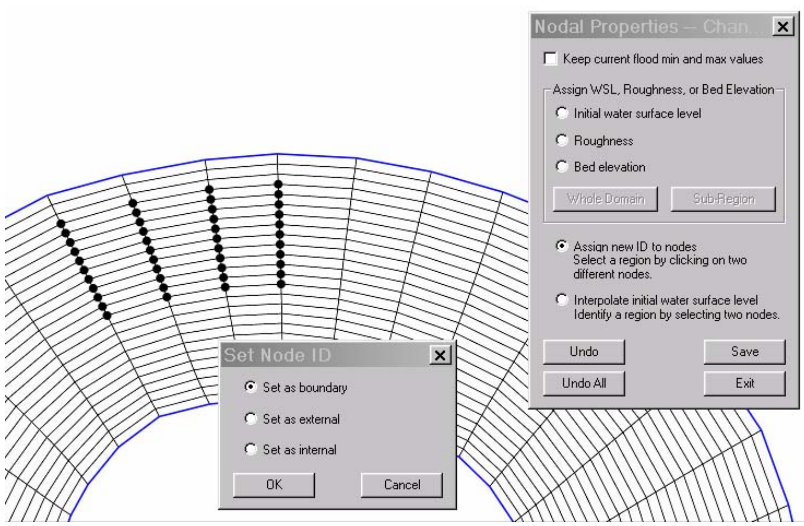


Figure 4-5

Three nodal types, namely, **internal nodes**, **boundary nodes**, and **external nodes**, are identified in CCHE2D model**.** You can change the nodal type for a specified region represented by a pair of diagonal points. For example, if you want to **construct an island** in the domain, follow the following steps:

- Click the radio button **Assign new ID to nodes** and then **select a region by clicking on two different nodes**.
- Check the option **Set as boundary** and click **OK**. The region will be surrounded by the blue lines.
- Click two diagonal points to specify the region covering all the other nodes within the blue lines.
- Check the option **Set as external** and click **OK**.
- **Warning: You must be very careful when changing the nodal type. It may result in an ill-defined mesh and the model may fail to run. Each time when you want to construct an island, make sure follow the above correct steps. You should always use the CCHE2D Mesh Generator to create high quality mesh.**



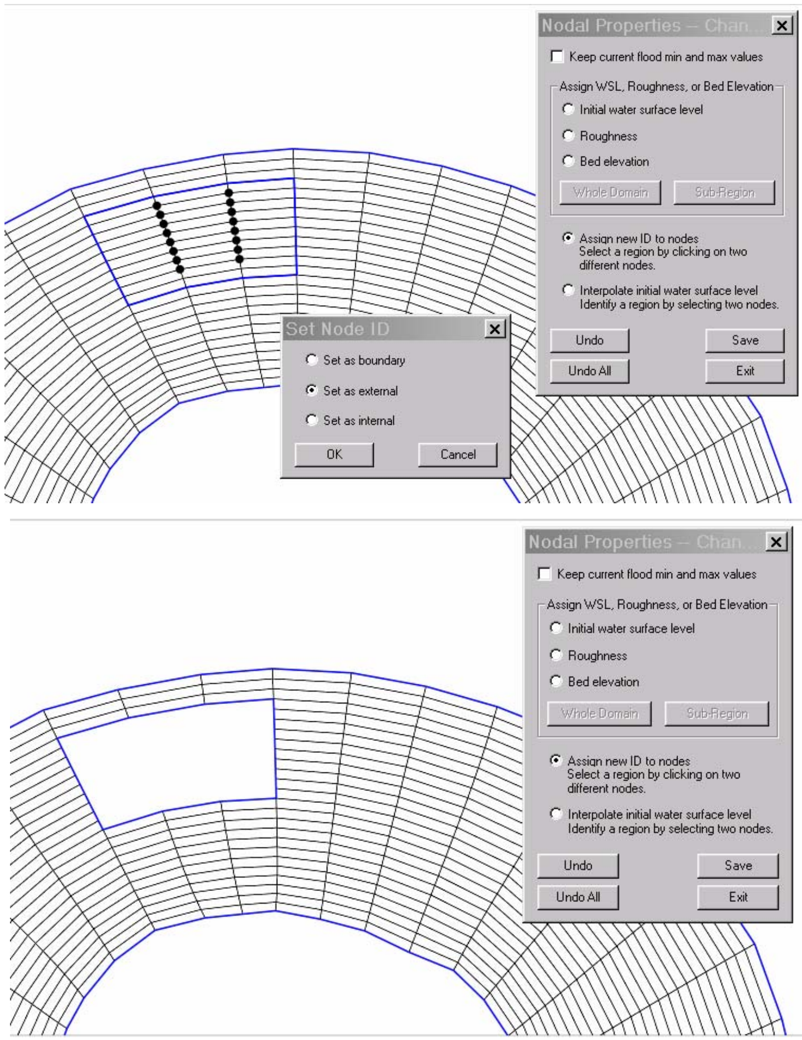


Figure 4-6

4.4 Set Flow Parameters

The flow parameters consist of **Simulation** parameters, **Bed Roughness** parameters and **Advanced** parameters. The **Set Flow Parameters** window can be invoked by clicking **Set Flow Parameters…**in menu **Simulation**.

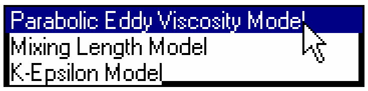
In page **Simulation Parameters,** the parameters are divided into five groups. Each parameter is explained as follows:

- In group **Time Step,**

- **Simulation time (s)**: the total time period in seconds of the simulation. In case of steady flow, this time should be sufficient so that a steady solution could be achieved
- **Time step (s)**: the step for time marching of the simulation.

- In group **Time Step for Output**, you can set the step intervals to output results into the **Intermediate File**, the **History File,** and for the **Monitor Points**, and set the step intervals for screen outputs indicating **Convergence** process.

- In group **Turbulence**, there are three **Turbulence Model Options**:



  - **For Parabolic Eddy Viscosity Model** and **Mixing Length Model**, you can set the **Turbulence viscosity coefficient**. This coefficient serves as a multiplier, i.e., a value of 10 means that the turbulent viscosity is 10 times that computed from the selected turbulence model. Although normally it equals 1, this coefficient has been tested in a range of [0.1, 1000].

- In group **Unsteady Flow Computation**, you can choose to **Compute as quasi-steady flow** instead of the real unsteady flow. If the flow boundary condition at any of the inlet section is unsteady, i.e., discharge hydrograph is specified, both the sediment and flow time steps are the same and the flow and sediment transport simulations are performed for each time step. However, for long-term simulation the computation time may be quite long. To increase the computation efficiency, the user has the choice to turn on the option that computes the unsteady flow as quasi-steady flow. In this case, the discharge hydrograph provided by the user must be a step function. The discharge during each step is considered as constant and the flow and sediment simulations are performed as for steady flow. Suppose (Q_i, T_i) and (Q_{i+1}, T_{i+1}) represents two consecutive hydrograph ordinates then for quasi-steady computation the discharge Q_i is assumed to be valid for the duration between T_i and T_{i+1}. In case of unsteady flow computation, the discharge is interpolated for any time between T_i and T_{i+1}. You can set the **Time steps to reach steady state** during each step. **Note that the time series in the hydrograph files of the inlets and outlets must be the same. The quasi-steady simulation cannot start from rest and must run from some base result.**

- In group **Numerical**,
  - **Wall slipness coefficient**: It is used to specify the wall boundary condition at no-flow boundaries. A value of 0.0 means no slip condition and a value of 1.0 means total slip, i.e., tangential velocity at no flow boundaries is allowed. A value between 0.0 and 1.0 would mean partial slip. A value greater than 1.0

signals the application of log-law. The log-law boundary condition allows partial slip, however, the shear stress is accurately predicted.

- **Depth to consider dry (m)**: It is used to determine the wet and dry nodes. The minimum water depth above it used to consider a node to be wet, while the water depth below that value would make a node dry.
- **Time Iteration Method**: This parameter provides information about the number of internal interations per time step. The **Method 1**, **Method 2** and **Method 3** mean small, medium, and large number of iterations per time step. The actual number is set by the computational model. The value should be based on the time step size, i.e., if the time step size is large the iteration control flag should be set to a higher value.

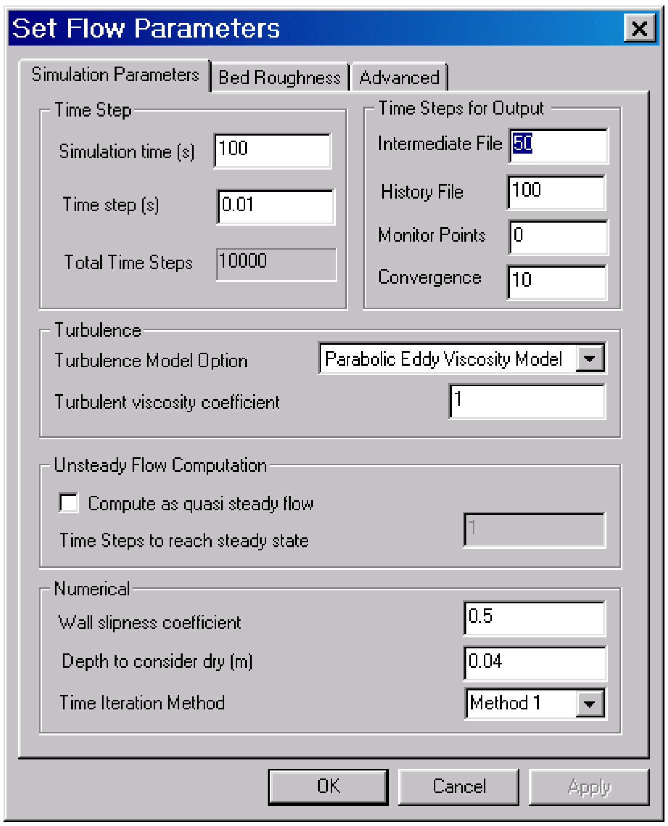


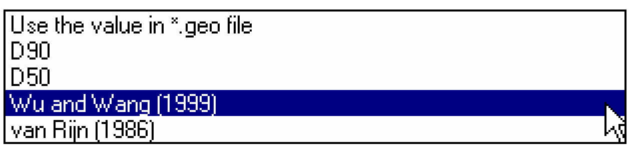
Figure 4-7

In page **Bed Roughness**, the parameters are divided into two groups: **For Flow Simulation Only** and **For Sediment Transport Simulation**.

- For flow simulation, you can choose to either **Use Values in .geo File** or **Use Bed Roughness Formula**. The roughness value in Geo file can be either **Manning's n** or **Roughness height Ks**. You must specify if the roughness values in the .geo file are Manning's n or roughness height.

  - Two bed roughness formulas are available for calculation: **Wu and Wang (1999)**; and **van Rijn (1986).** If you choose roughness formula for flow, the selected formula will be also used in the bed roughness calculation for sediment. You also need to specify the formula parameters, such as **D16**, **D50**, **D90** and **Calibration Factor**. The **Calibration Factor** is within the range of **[0.2, 5.0]** and its default value is **1.0**

- For sediment transport simulation, there are five methods to calculate the bed roughness. You can **Use the value in *.geo file**, or calculate bed roughness according to the sediment diameter size of **D90** or **D50**, or use **Wu and Wang (1999)**'s formula and **van Rijn (1986)**'s formula**.**



- The options **Wu and Wang (1999)** and **van Rijn (1986)** can be set in either group **For Flow Simulation Only** or **For Sediment Transport Simulation**. Setting them in one group equals to setting them in both groups at the same time.

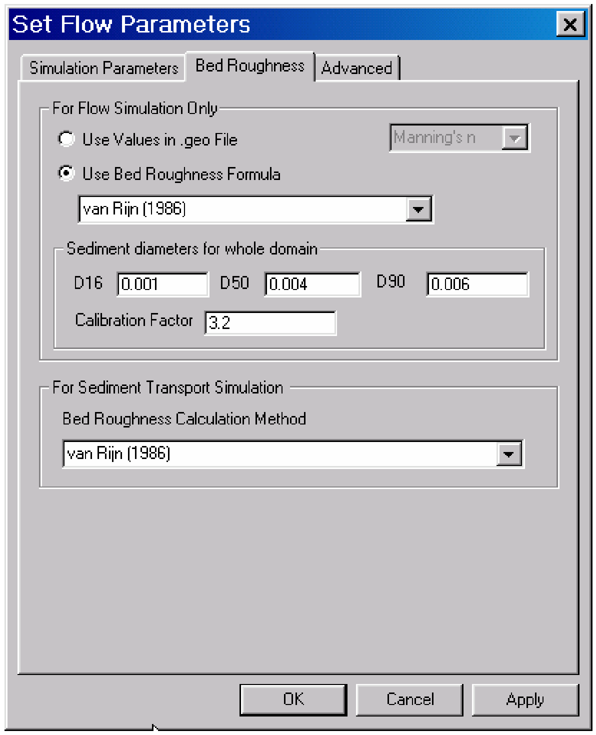


Figure 4-8

In page **Advanced**, there are four parameters, namely, **Coriolis force coefficient**, **gravitational acceleration**, **von Karman constant**, and **kinematic viscosity of fluid**, with default values that suffice for most cases, however, if needed you can change the default values.

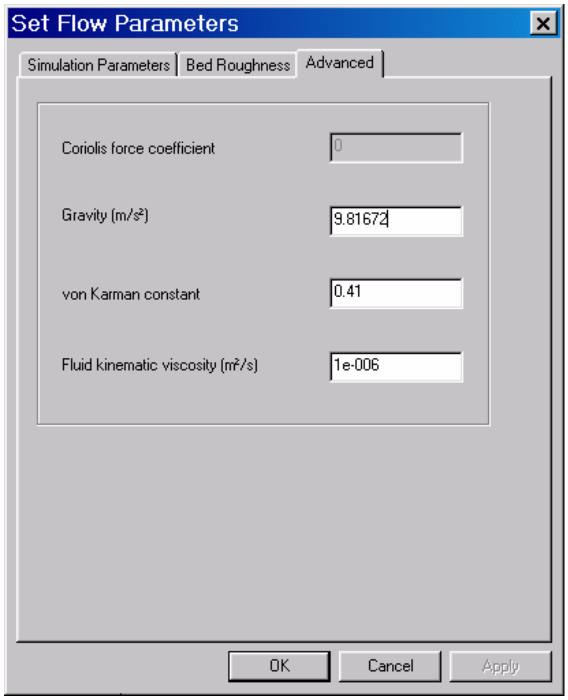


Figure 4-9

4.5 Set Sediment Parameters

To run sediment transport simulation, you must specify the sediment parameters and the initial and boundary conditions. The basic procedure of setting sediment information is listed as follows:

- Define the number of layers.
- Define the sediment size classes. All bed material and boundary condition data are specified according to these classes.
- Specify the sediment boundary conditions.
- Define the bed material samples.
- Set the bed material properties.

Sediment parameters are specified through the **Set Sediment Parameters…** in the **Simulations** page. There are four pages, **Sediment Size Classes**, **Sediment Transport**, **Sediment**, and **Bed Roughness**.

In page **Sediment Size Classes**,

- **Number of Bed Layers**: is used to define the bed configurations in vertical direction. The default value is 3. The first top layer is the mixing layer where the exchanges between sediments in water and on bed occur.
- **Minimum Mixing Layer Thickness**: is a numerical parameter used to confine the bed erosion process.
- In group **Define Size Class**,
  - To add a size class, input the sediment size diameter one at a time and click **Add Size Class.**
  - To delete a size class**,** select a sediment size in **Mean diameter (m) of each size class** and click **Delete Size Class.**
  - To delete all size classes, click **Clear All**.
  - **Notes: The maximum number of sediment size classes is 8. Practically this number should not be more than 5, otherwise the computational efficiency will be reduced significantly.**

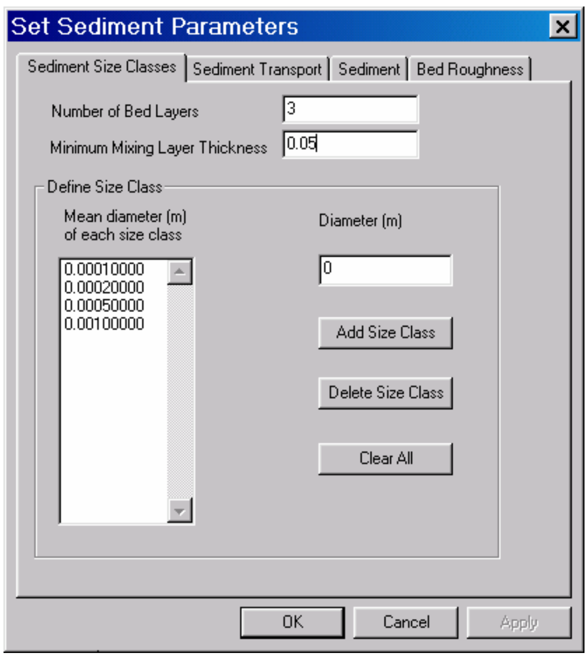
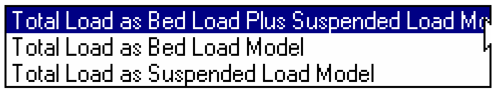
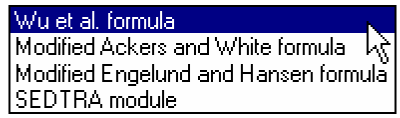


Figure 4-10

In page **Sediment Transport**, four groups of parameters need to be specified.

- In the first group, you can set the **Transport Mode** and the **Transport Capacity Formula**. There are five transport modes and four capacity formulas available.





  - **Total Load as Bed Load Plus Suspended Load Model** is for simulation of both the bed load and the suspended load (total load).
  - **Total Load as Bed Load Model** is for simulation of both the bed load and the suspended load but **with the bed load dominant**.
  - **Total Load as Suspended Load Model** is for simulation of both the bed load and the suspended load but **with the suspended load dominant**.
  - If you select **Total Load as Bed Load Model** or **Total Load as Suspended Load Model**, you need to choose appropriate sediment **transport capacity formula.**
- In group **Sediment Simulation Mode**, two simulation modes are available. The first option **Slow Bed Change** can be selected only if the flow is steady, while the option **Fast Bed Change** is for unsteady flow. Note that if you choose to **Compute as quasi-steady flow** in page **Simulation Parameters** of **Set Flow Parameters** (please refer to section 4.4 for details), it implies the simulation mode of Fast Bed Change for sediment transport and you will not be able to set other simulation mode.
- In group **Adaptation Length for Bedload**, you can choose appropriate option to calculate adaptation length for bed load for non-equilibrium sediment transport.
- In group **Adaptation Length Factor for Suspended Load,** similarly, you can choose appropriate option to calculate the adaptation length factor for suspended.
- The non-equilibrium adaptation length characterizes the distance for sediment to adjust from a non-equilibrium state to an equilibrium state. It is a length scale for the river bed to respond the disturbance of the environment, such as hydraulic structures, channel geometry changes and incoming sediment variation.

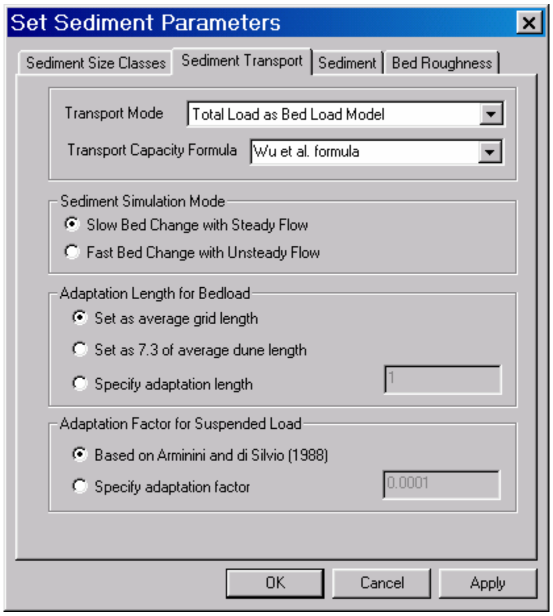


Figure 4-11

In page **Sediment**,

- The **Sediment specific gravity** has a default value of 2.65 that suffices for most applications.
- The **Curvature Effects** can be included into the sediment transport simulation if the domain has the curved parts. You need to set the **Average channel width** accordingly.
- In group **Steady Flow Computation**,
  - The **Time steps to adjust flow** is used to adjust the flow after the bed changes. The flow will be recalculated for the number of time steps specified by the user.
  - The **Erosion/Deposition limit (0.01-0.05 of depth)** is used to restrict the maximum amount of erosion/deposition in the domain within a time step. If erosion/deposition at any node within a time step exceeds the limit specified by the user, the time step is reduced and computations are repeated.
  - The above parameters are effective only if steady flow boundary conditions are prescribed at all the inlets.

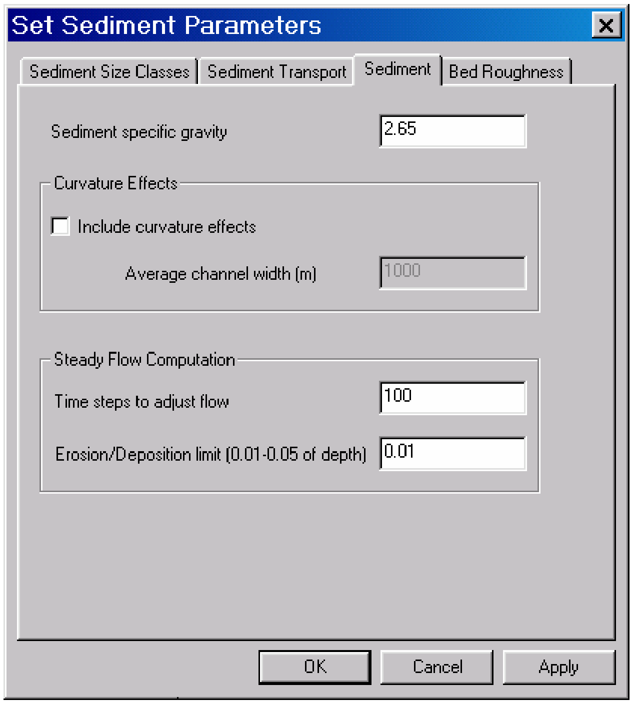


Figure 4-12

The page **Bed Roughness** is the same as the page **Bed Roughness** in **Set Flow Parameters**. Please refer to section 4.4 for details.

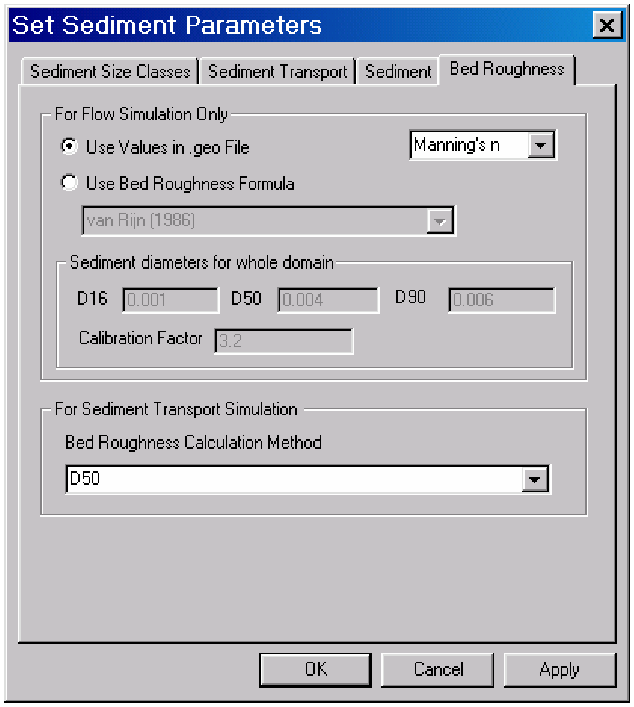
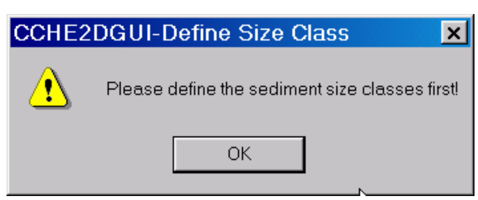


Figure 4-13

4.6 Set Sediment Boundary Conditions

The information of sediment boundary conditions are provided by the suspended sediment boundary condition file (***.sbc**) and the bedload boundary condition file (***.bbc).** The CCHE2D-GUI provides a file editor to help you edit these two files.

- To edit these two files, select **Set Sediment Boundary Conditions…**in menu **Simulations**. Please make sure you already defined the sediment size class, otherwise an error message will appear.



- In the **File Editor**, you can create (**New**), **Import** and **Save** the sediment boundary conditions file selected from the **File Selector**.

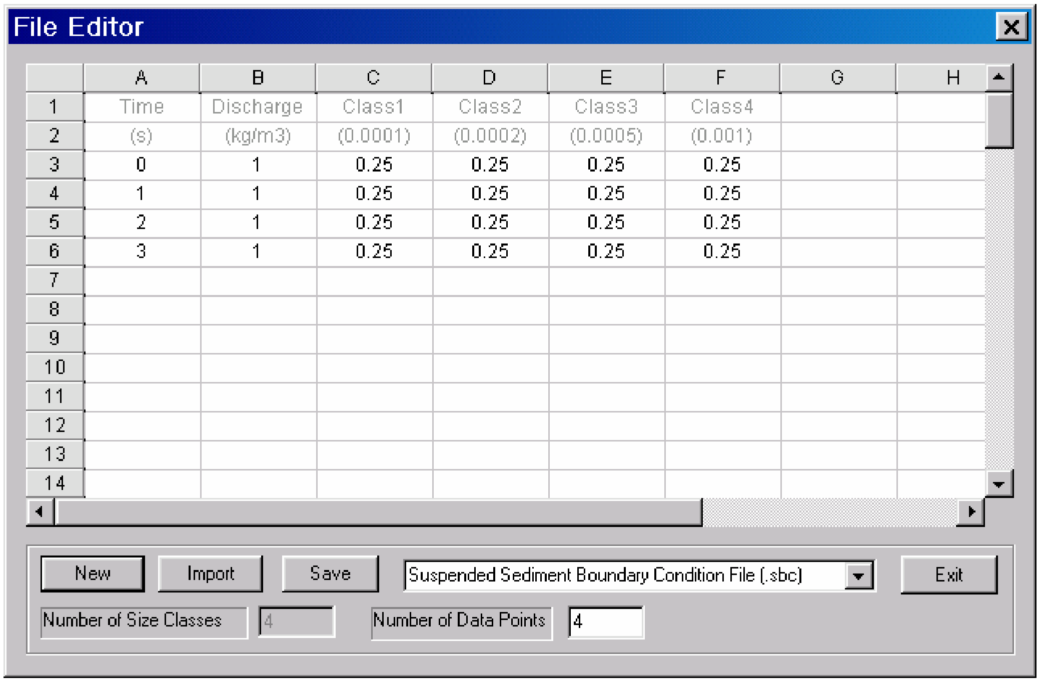


Figure 4-14

- To import an existing selected file, click **Import.**
- To save the changes to a selected file, click **Save.**

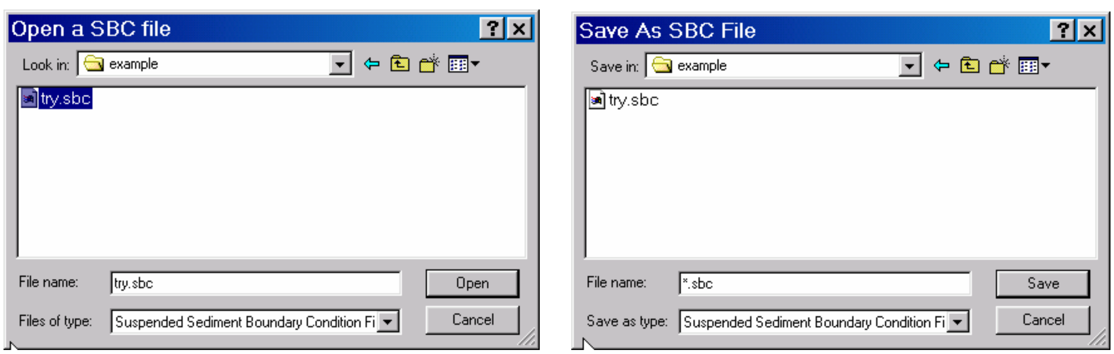


Figure 4-15

- To create a template for the selected file, click **New.** The **sbc** and **bbc** template is created based on the **Number of Size Classes** and **Number of Data Points.** The **Number of Size Classes** is a predefined value and you cannot set it here. The **Number of Data Points** defines the number of time series points. The **sbc** and **bbc** files have the same format. The unit of **Sediment Discharge** in **sbc** is kg/m^3 while in **bbc** is kg/s.

- To edit the cell value, click the active cell and then type the desired values. The cells in gray color are inactive. You can **Cut**, **Copy**, **Paste** and **Delete** by right clicking the selected cells to invoke the popup edit menu.

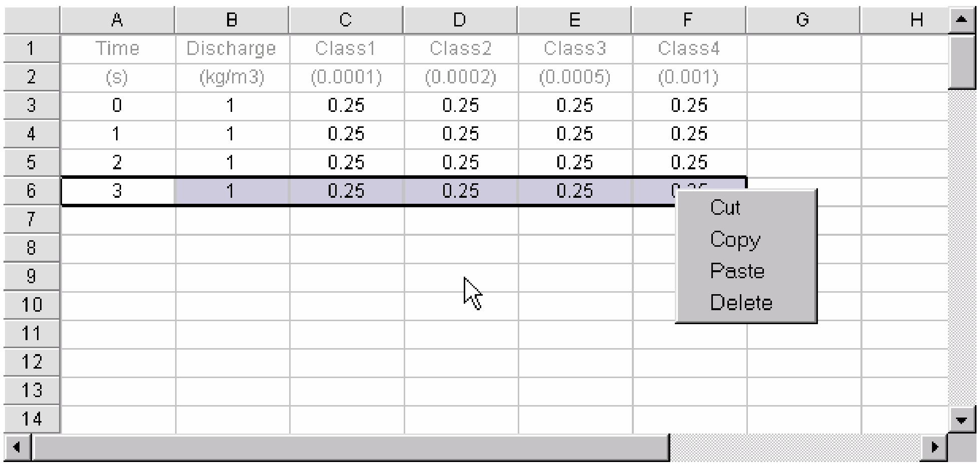


Figure 4-16

- Note that once the **Number of Data Points** is set, you cannot add more data points by directly inputting values into cells. The sum of the factions of the size classes at each time series should be 1, other wise an error message will appear.

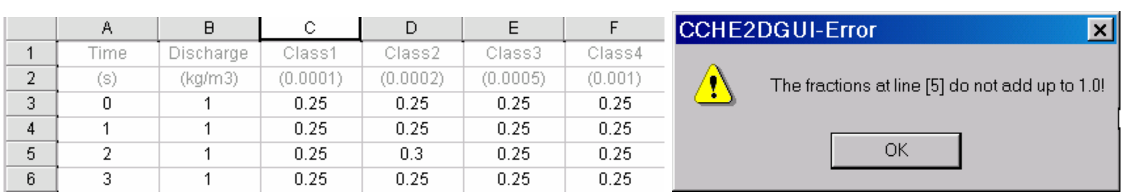
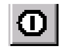


Figure 4-17

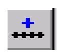
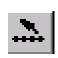
4.7 Set Inlet/Outlet Boundary

To set inlet or outlet boundary conditions, there are two steps to follow: editing boundary node strings and attaching boundary conditions to the node strings. You need help from the tools of editing boundary conditions which can be activated by clicking **Start Editing**

Inlet/Outlet Boundary from menu **Simulation** or button  on the boundary editing toolbar .

There are two kinds of boundary node strings, **inlet boundary node string** (denoted by an arrow entering the domain), **outlet boundary node string** (denoted by an arrow going out of the domain).

Figure 4-18

- To **add a boundary node string**: click  and then click two different points along a constant I line or J line. Make sure the node string is **at least three nodes wide.**
- To **delete a boundary node string**: click  and then click the node string you want to delete.
- To **modify a boundary node string**: click  and then click a point on the I line or J line on which the node string is located. The length of the node string will be changed. You also need to make sure the modified node string is **at least three nodes wide.**
- To **select a boundary node string**: click  and then click the desired node string. If it is inlet boundary node string, you need to attach the **Inlet Boundary Conditions;** if it is an outlet boundary node string, you need to attach the **Outlet Flow Boundary Conditions;** and if it is a newly added node string, you need to determine its boundary type in **Select Inlet/Outlet Boundary.**

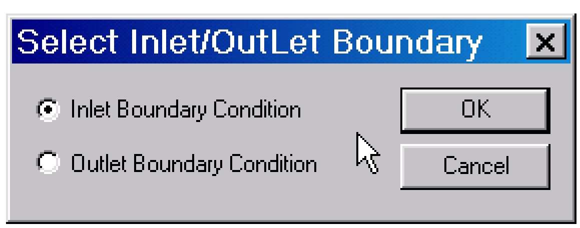


Figure 4-19

For **Inlet Boundary Conditions**, you need to set the flow inlet boundary conditions and the sediment inlet boundary conditions.

- In page **Flow**,
    - You can specify either **Total discharge** or **Discharge hydrograph** with **Inflow angle (degree)** at the inlet. If the **Discharge hydrograph** is selected, you need to provide a discharge hydrograph file (***.dhg**) whose file format can be found in **Appendix. Note that the hydrograph must start from t = 0.**
    - A shortcut of **Set Flow Parameters…**is provided here. For details, please refer to section 4.4.
    - To detach a flow boundary condition, click **Delete Flow BC**.

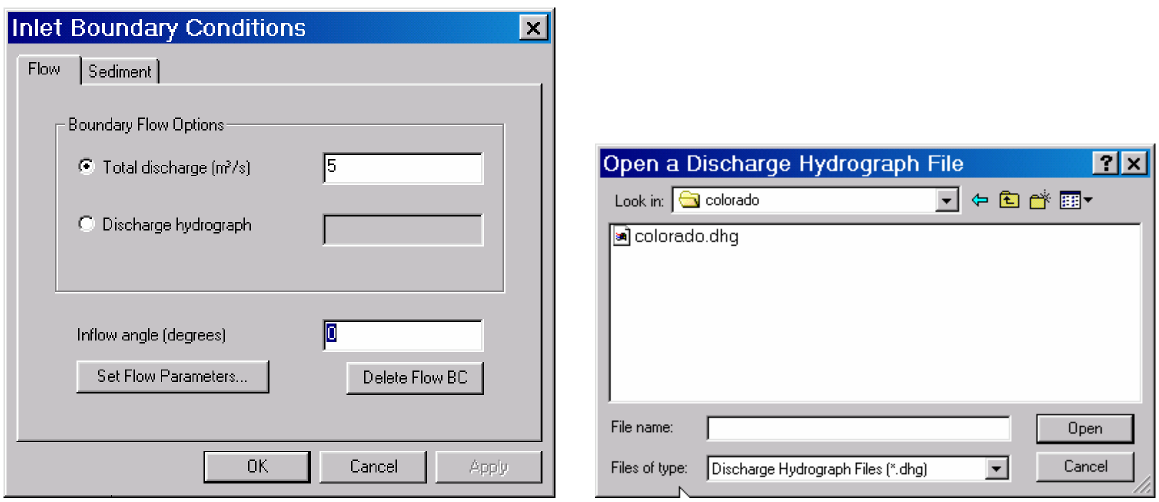


Figure 4-20

- In page **Sediment**, you need to provide a bed load boundary condition file (***.bbc**) or a suspended load boundary condition file (***.sbc**) or both, which depends on the option you selected in the **Transport Mode** of page **Sediment Transport** in **Set Sediment Parameters**.
    - To attach an existing sediment boundary condition, click **Import BC File…** in group **BedLoad Sediment** or **Suspended Sediment**. If no existing sediment boundary condition is available, you can create a new sediment boundary condition by clicking **Create BC File….**and the **File Editor** will appear. For details on how to edit sediment boundary conditions using **File Editor**, please refer to section 4.6.

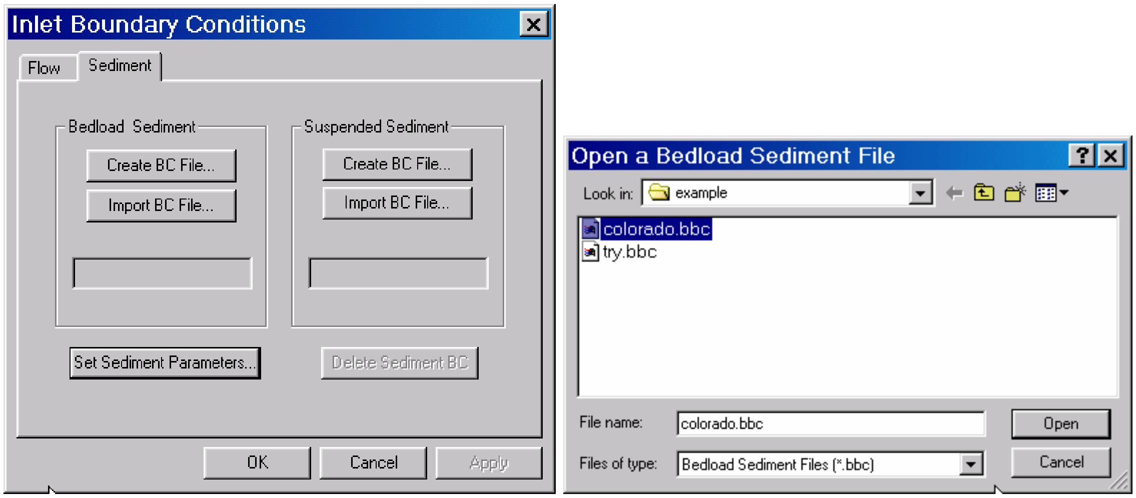


Figure 4-21

- To detach a sediment boundary condition, click **Delete Sediment BC**.

For **Outlet Boundary Conditions**, four kinds of outlet flow boundary conditions are available, **Open boundary conditions**, **Water surface level**, **Rating curve** and **Stage Hydrograph.**

- In case of **Open Boundary Condition**s, the model will estimate the water surface level at the outlet boundary based on kinematic wave condition. This option is valuable if you cannot assess the water level at the channel outlet.
- If you select **Rating curve**, you need to provide a rating curve file (***.rcv**) with stage-discharge relationship. Please see the Appendix for instructions.
- If you select **Stage hydrograph**, you need to provide a stage hydrograph file (***.shg**) with time-stage relationship. **Note that the stage hydrograph must start from t = 0.**

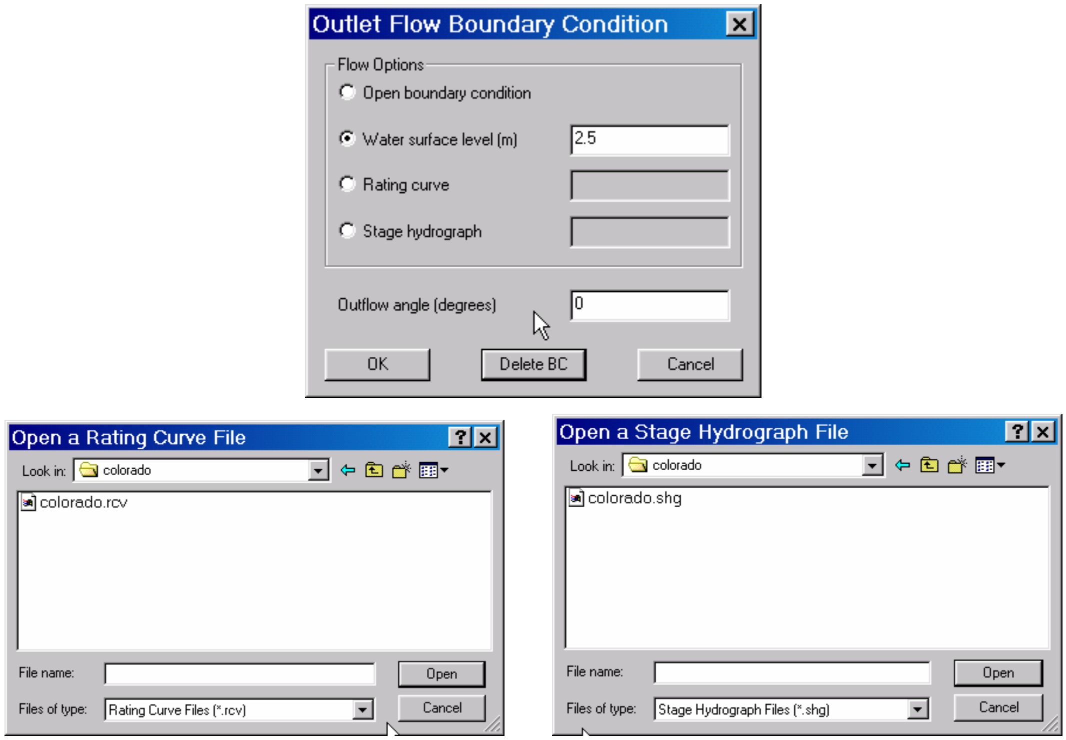
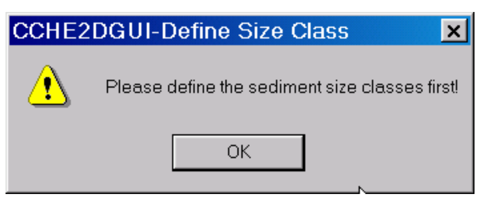


Figure 4-22

4.8 Set Bed Material Samples

The bed material samples will be used to define the initial bed material compositions in both horizontal and vertical directions for the entire domain. The information of the bed material samples are contained in a file with the extension **bmt.** To set bed material samples, select **Set Bed Material Samples…** in menu **Simulation**. Please make sure you already define the sediment size classes, otherwise an error message will appear.



In **Define Bed Material Samples**,

- If the **bmt** file exists, the defined bed samples will be displayed, otherwise a blank sheet is displayed.
- You need to set the **Porosity** and the fractions of each sample. You don't need to set the **Sample No**.

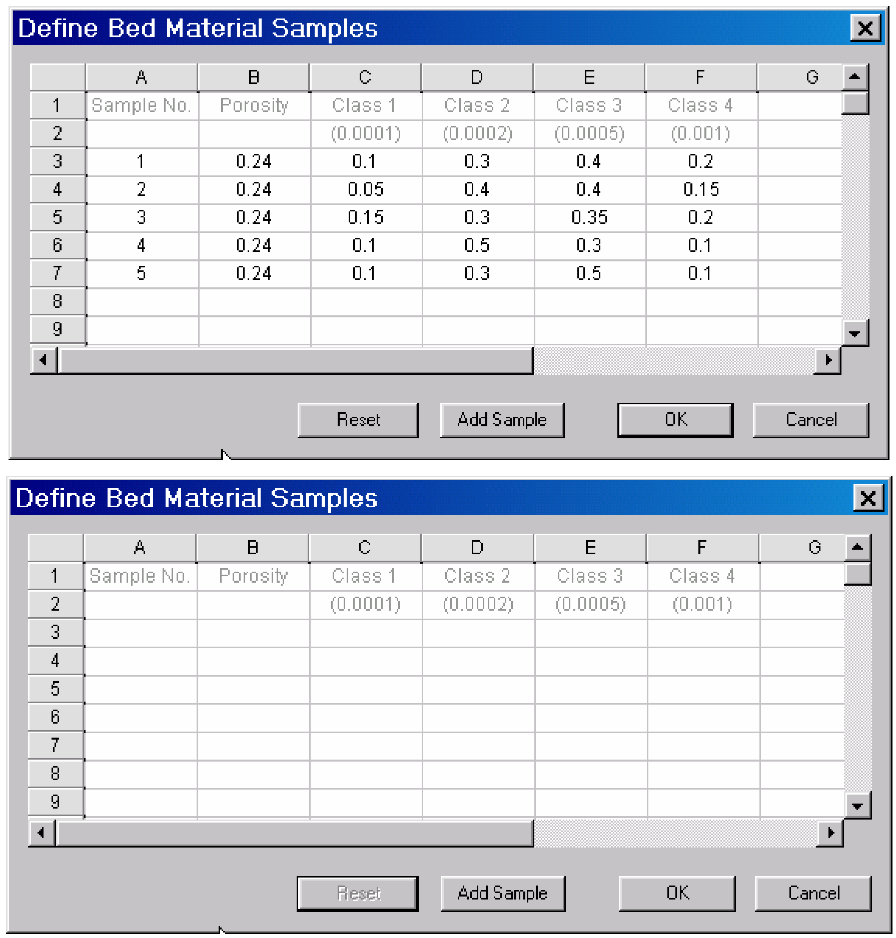


Figure 4-23

- To create a new sample, click **Add Sample**. A sample with default porosity and equal-distributed factions will be added at the end of the records.
- To edit a sample, click the corresponding cell and type the desired value. Please make sure the sum of the factions of each sample be 1, otherwise an error message will appear.

- To save the samples, click **OK**. The defined bed samples will be saved into the **bmt** file and a **bed** file which defines the initial bed material composition for each mesh node will be created at the same time using the information from the **bmt** file. If the **bed** file is already existing, you need to confirm if it is overwritten or not.

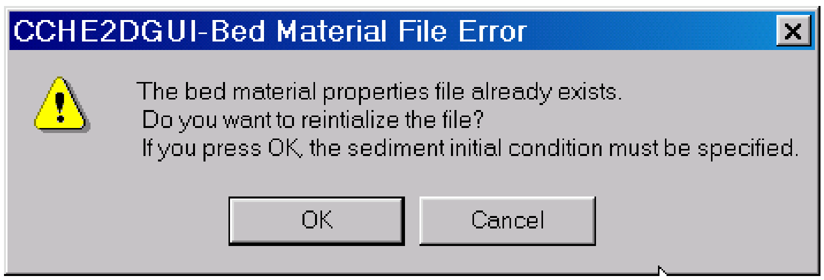


Figure 4-24

- To delete all samples, click **Reset**.

4.9 Set Bed Material Properties

After the bed samples are defined, you can set the initial bed material properties contained in a file with the extension **bed**. To set the bed material properties, select **Set Bed Material Properties…**in menu **Simulation**. If the information in **bmt** file and **bed** file is not compatible, you can create a new **bed** file based on the information from **bmt** file.

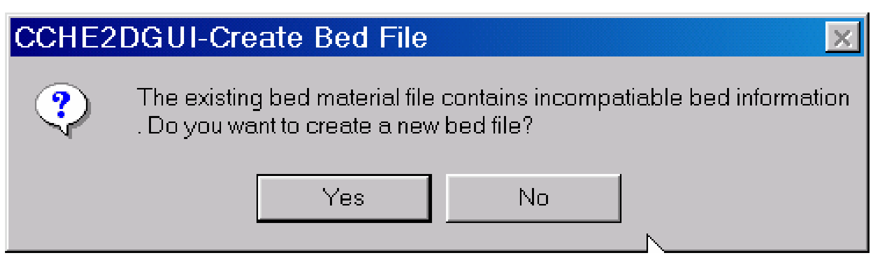


Figure 4-25

There are five properties divided into two groups for each mesh node, **Layer Thickness**, **Bed Sample Number**, **Erodibility**, **Maximum Erosion Thickness**, and **Maximum Deposition Thickness**.

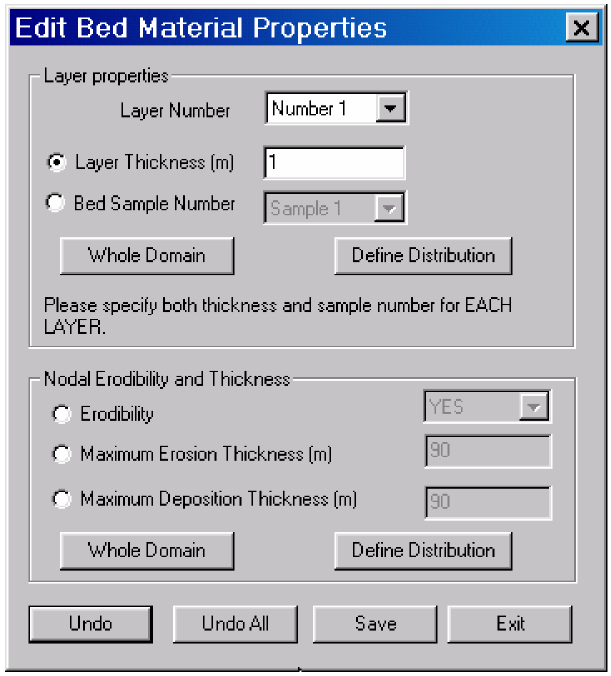


Figure 4-26

In group **Layer properties**,

- You must set the layer thickness and layer sample number for each layer. The **Layer Number** is set in page **Sediment Size Classes** in **Set Sediment Parameters** (please refer to section 4.5). The samples are set in **Define Bed Material Samples** (please refer to section 4.8).
- To set the layer thickness or layer sample number, type the desired value or select desired sample, then click **Whole Domain** or **Define Distribution**. If you click **Whole Domain**, the selected property will be assigned a constant value for each node. You can also give a spatial distribution to the selected property by clicking **Define Distribution**. After clicking this button, the GUI expects you to define the region by **selecting two points** which are the diagonal points of a rectangular area within the mesh.

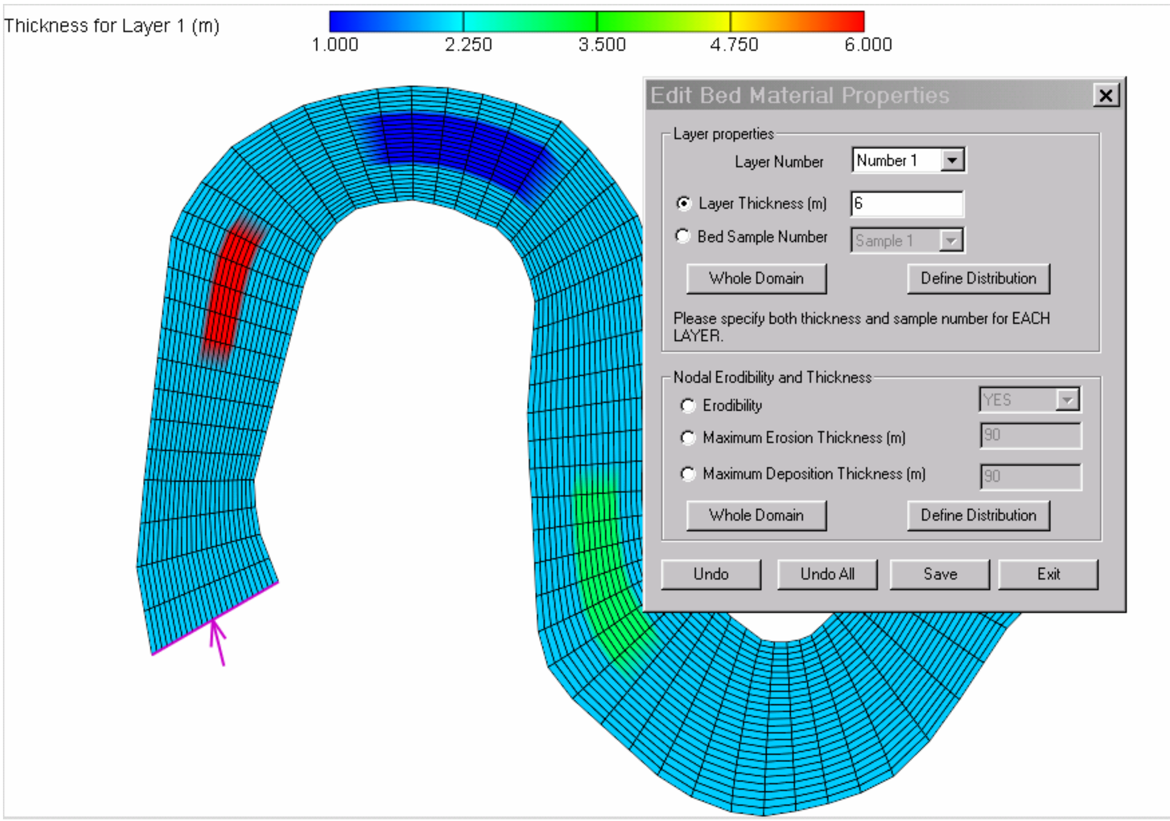


Figure 4-27

Note that you need to specify the layer properties for EACH LAYER.

In group **Nodal Erodibility and Thickness**,

- The erodibility helps differentiate the non-erodible and the erodible nodes. For example, the nodes on the concrete hydraulic structure should be non-erodible.
- The **Maximum Erosion Thickness** and **Maximum Deposition Thickness** are used to limit the erosion and deposition process, respectively. By default, a uniform value of 90 meter is used, that is, no limit is enforced.
- For a selected property, you can set a constant distribution by clicking **Whole Domain.** You can define the distribution by clicking **Define Distribution**. After clicking this button, the GUI is waiting for you to define the region by **selecting two points** which are the diagonal points of a rectangular area from the mesh.

To save your settings, click **Save**. The changes will be saved into the **bed** file.

To undo the changes of the previous step, click **Undo**.

To undo all the changes, click **Undo All.**

4.10 Set Monitor Points

Monitor points are used to get history of results at selected nodes. The current version of the CCHE2D model allows up to five monitor points in the computational domain. As the CCHE2D model is an unsteady flow model, even the steady state is attained through forward marching in time. Thus monitor points are valid both for the steady and unsteady flow simulation. To set the monitor points, select **Edit Monitor Points…** in menu **Simulation**.

- To add monitor points through IJ Indices, input the **I index** and **J index** and then click **Add Point** in group **Monitor Point Indices.**
- To Add monitor points through Coordinates, input **x coordinate** and **y coordinate** and then click **Add Point** in **Monitor Point Coordinate**. A grid point closest to this location (x, y) will be added.
- You can also click **Add Points from Grid** and then click a grid point.
- You can also click **Select Monitor Point** and then click the desired place on the mesh. A grid point closest to this point will be added as a monitor point.
- You can **Remove Current Point** and **Remove All Points**
- To save the monitor points, Click **Save.**
- If you set the monitor points, you need to set time step for monitor points output on page **Simulation** in **Set Flow Parameters**.

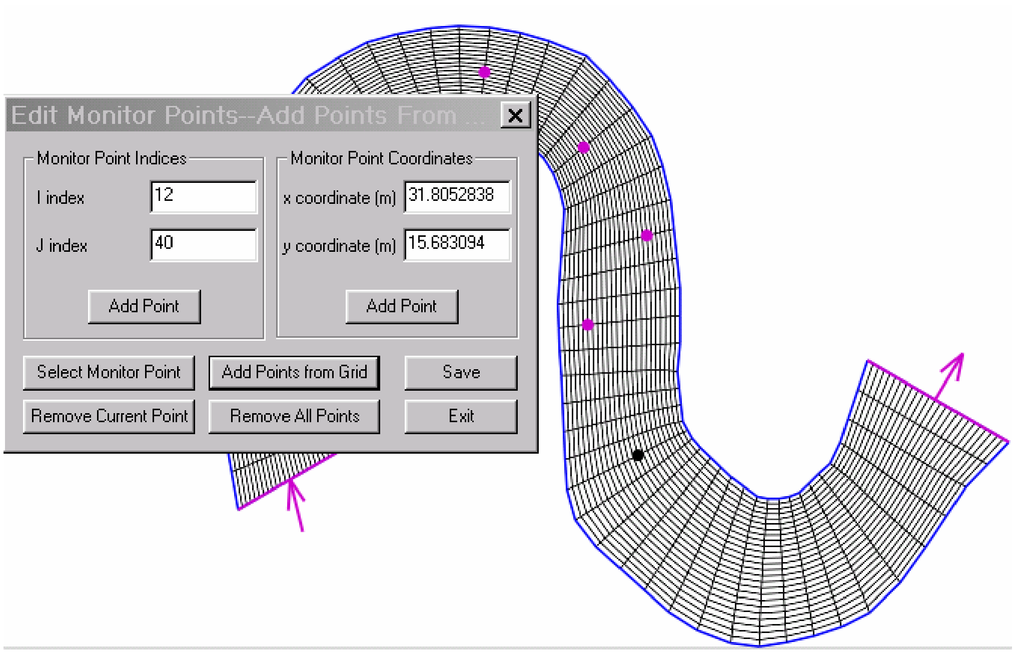


Figure 4-28

4.11 Run CCHE2D Model

After all the initial conditions and the boundary conditions are set, the simulation can be performed. To run CCHE2D model locally, select **Run CCHE2D Model…** in menu **Simulation** or click  in the main toolbar.

In **Simulation Options**, the current **Run Number** is displayed at the top of the window. This **Run Number** will be used to name the corresponding results file. The flow result file has the extension "**flw**", while the sediment result file has the extension "**sdm**". The name of the result file has the following generic form:

*CaseName*_Run-*Current Run Number* (*Start Run Number*).flw (or sdm)

For example, if the *CaseName* is "try", the *Current Run Number* is 4, and the *Start Run Number* is 1, the result file will be: "try_Run-4(1).flw" or "try_Run-4(1).sdm". The *Start Run Number* represents the results from the corresponding run. If it is 0, it means the simulation will begin from rest.

There are four simulation options.

- **Start Flow Simulation from Rest**: It is also called "cold start" for flow simulation. The flow simulation will begin with the initial water surface specified as in section 4.3 and the initial velocity field (velocities are set to zero by default)

.

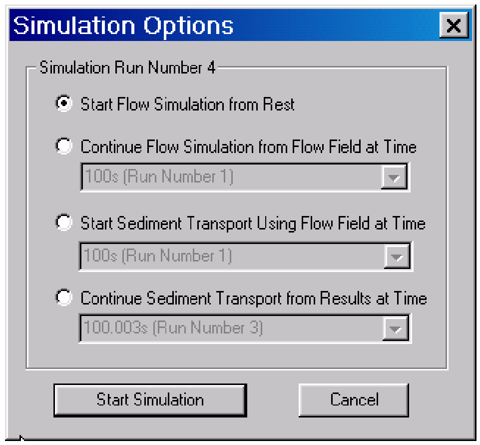


Figure 4-29

- **Continue Flow Simulation from Flow Filed at Time**: It is also called "hot start" for flow simulation. The flow simulation will begin with the computed flow field at a selected time. You can select the flow field from the corresponding flow filed selector.

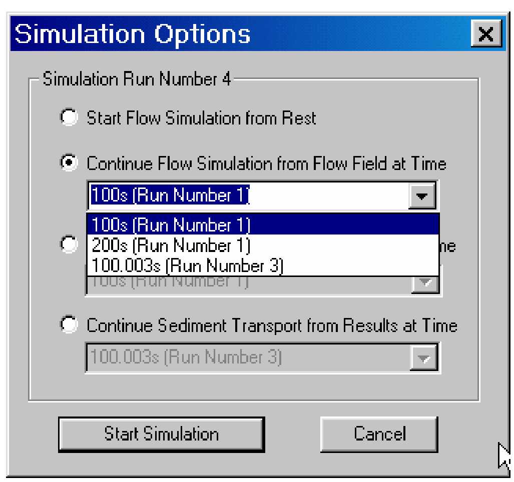


Figure 4-30

- **Start Sediment Transport using Flow Field at Time**: It is also called “cold start” for sediment transport simulation. The sediment transport will begin with the initial bed defined in the mesh file (**geo**) and the computed flow field at a selected time. You can select the flow field from the corresponding flow field selector.

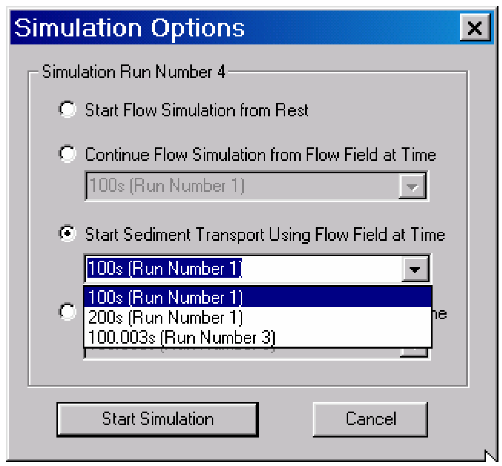


Figure 4-31

- **Continue Sediment Transport from Results at Time**: It is also called “hot start” for sediment transport simulation. The sediment transport will begin with the computed bed and flow field at a selected time. You can select it from the sediment results file selector. Use this option to continue a sediment transport simulation.

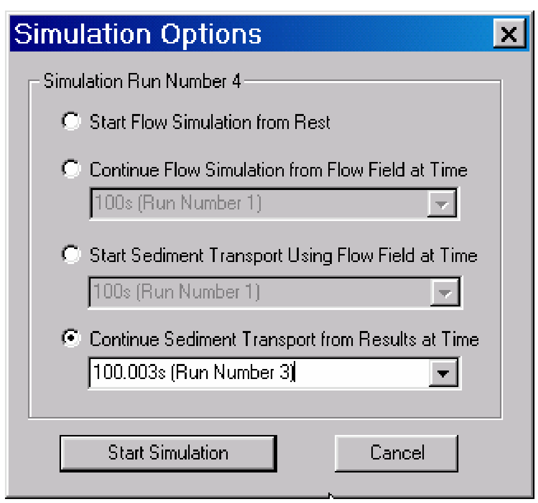


Figure 4-32

To actually start the simulation, first select the simulation option and then click **Start Simulation**. The GUI will check all the model parameters, and the initial and boundary conditions. After the check, the simulation will be carried out in a console window. Progress of the run is displayed in this window. You can close the CCHE2D-GUI while the model is running. Closing the console window will interrupt the simulation. During the simulation, you can visualize the intermediate results.

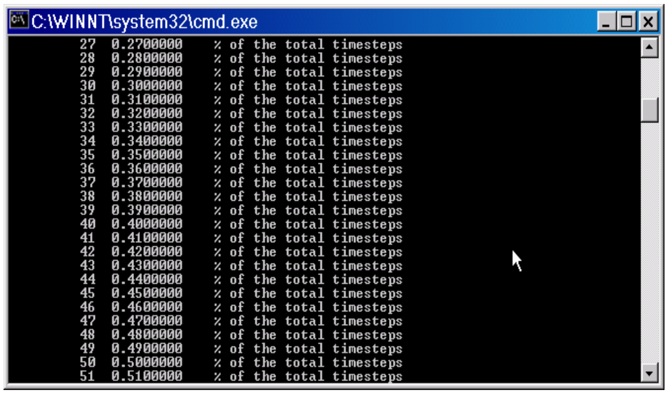
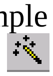


Figure 4-33

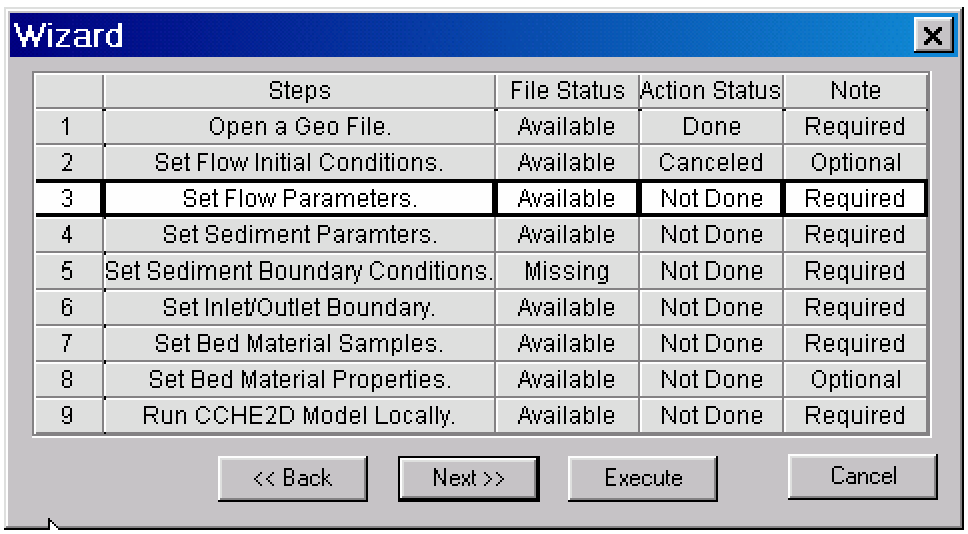
4.12 Use Wizard

To help the first-time users quickly master how to run simulations using CCHE2D-GUI, a simple wizard is provided. To use the wizard, select **Wizard** in menu **Simulation** or click  in the main toolbar.

In **Wizard**, only the necessary steps are listed. For each step, the **File Status**, **Action Status**, and **Note** are provided.

- The **File Status** shows the status of files associated with the corresponding step. "**Available**" denotes the associated files already exist, while "**Missing**" denotes the non-existing files that should be created by the CCHE2D-GUI or provided by the users.
- The **Action Status** traces the users' action on the corresponding step. "**Done**" means the corresponding step has been successfully executed; "**Canceled**" means the corresponding step has been executed but not completed; and "**Not Done**" means the corresponding step has not been executed or has been skipped.

- The **Note** shows the current properties of the steps. "**Required**" means the corresponding step must be executed, while "**Optional**" means the corresponding step may not necessarily be executed and can be skipped if not needed for the desired simulation.
- There are 9 steps, among which steps 3, 4, 5, 7, and 8 are related to the sediment transport simulation. The properties of these steps are determined by the sediment **Transport Mode** in page **Sediment Transport** of **Set Sediment Parameters** (please refer to section 4.4 for details). If a certain transport mode is selected, these steps become "**Required**" if you want to run sediment transport simulation. For flow simulation, you can skip these steps (3, 4, 5, 7 and 8).



.Figure 4-34

The navigator buttons **<<Back** and **Next>>** are used to select a step. Once a step is selected, click **Execute** to perform this step. Details on these 9 steps can be found in previous sections in this chapter.

5 Visualize Results

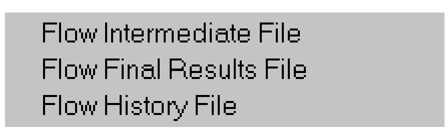
5.1 Introduction

Visualization of modeling results is another main function of the CCHE2D-GUI. It provides quite a few ways to visualize the results, such as flood color shading, contour lines, color points, vector plotting, and etc.

In CCHE2D-GUI, there are three visualization modes, namely, **Grid**, **Flow** and **Sediment**. In Grid mode, only the geometry file (***.geo**) is loaded; in Flow mode, the geometry file and a result file (***.mdw** or ***.flw** or ***.his**) are loaded; and in Sediment mode, the geometry file and a result file (***.mds** or ***.sdm** or ***.sed**) are loaded.

5.2 Visualize Flow Results

To visualize the flow results, first you need to load the result files by visiting menu **Visualization**. For flow, there are three kinds of result files available.



←(***.mdw**)
←(***.flw**)
←(***.his**)

Figure 5-1

5.2.1 Visualize Intermediate and Final Results

For flow intermediate file, you can choose to load it manually or automatically.

- To load it manually, select **Flow Intermediate File** in menu Simulation.

- To load it automatically, select **Auto-check Flow Intermediate Result**. You can set the time interval to check the result.

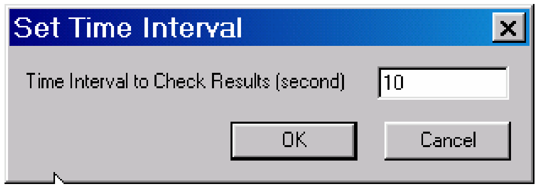


Figure 5-2

For **Flow Final Results File**, you need to further select the computed flow field at a certain time.

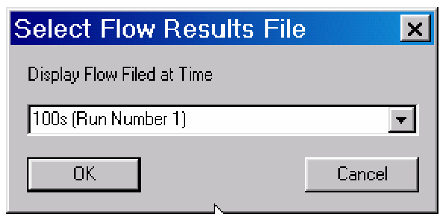


Figure 5-3

After a result file (**mdw** or **flw**) is loaded, you can select flow variables from **variable selector** on the main toolbar.

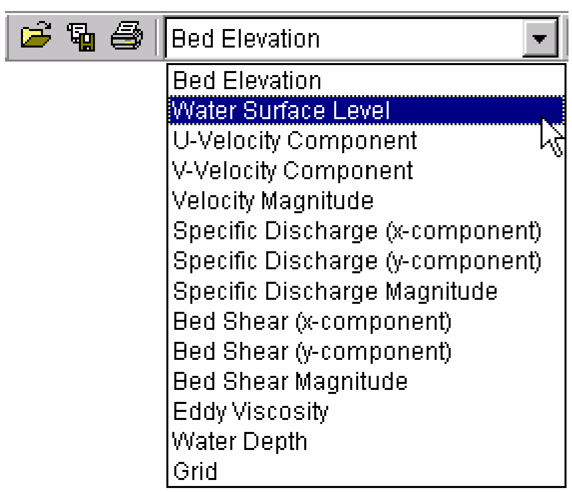


Figure 5-4

When reloading the results files (***.mdw** or ***.str**), you will be asked to set the **Results Settings**. You can keep the **flood scale settings**, **contour scale settings**, or **view coordinates** as the same as or different from the previous file loading.

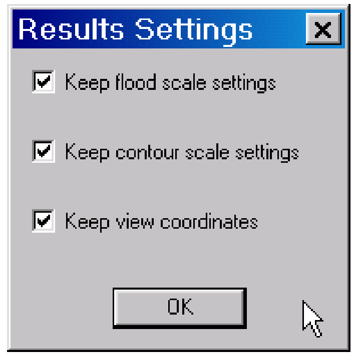


Figure 5-5

5.2.2 Visualize History File

For **Flow History File**, the **History File Editor** will be used to manipulate the visualization of the results.

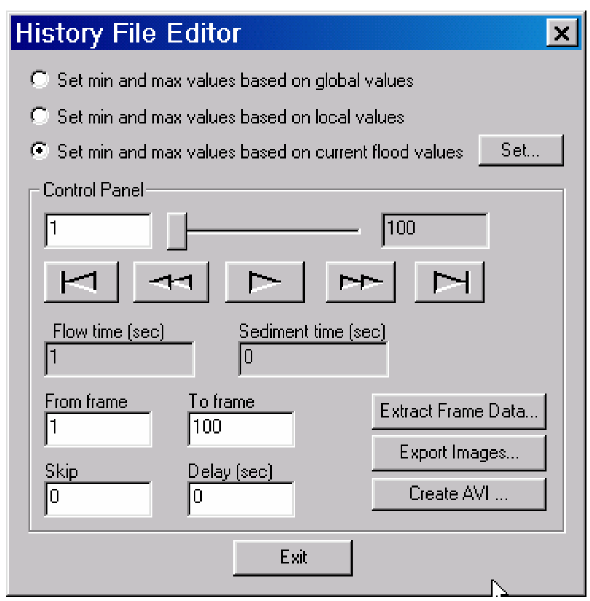


Figure 5-6

In **History File Editor,** there are two groups. In the first group, you can set the plot options of the min and max values for the current plot variable. If the third one is selected, you can further set the properties of the current flood values.

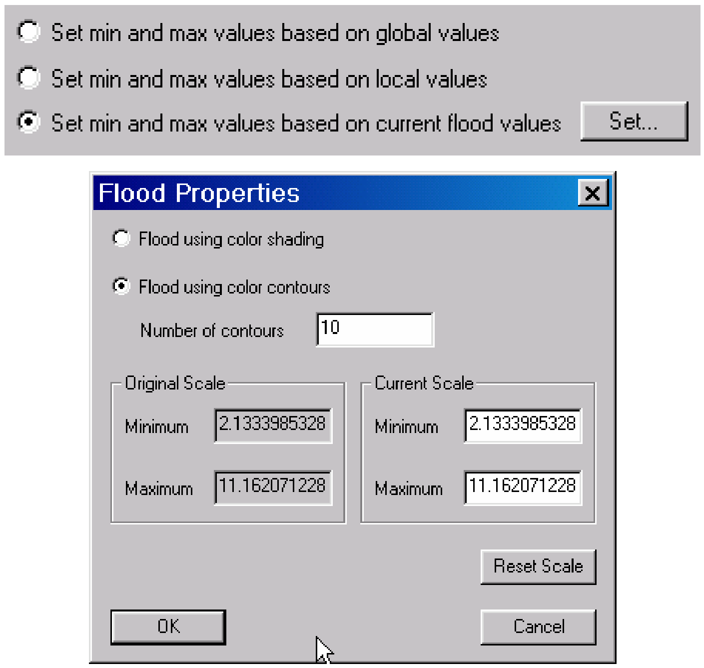


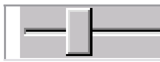
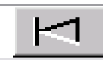
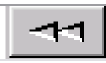
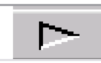
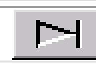
Figure 5-7

The second group is the control panel. The buttons related to frame operation are listed in the following table and others are explained as follows.

- **Flow time** and **Sediment time:** To display **t**he flow time and sediment time of the current frame.
- **From frame** and **To frame:** To set a range of frames.
- **Skip:** To skip a certain number of frames for editing.
- **Delay**: To set the interval time when playing frames.
- **Extract Frame Data**: To extract the data of the current frame and save it as the final result file (***.flw**).
- **Export Images**: To export the frames as a series of Bitmap files with a range set in **From frame** and **To frame.**

- **Create AVI…:** To create an AVI file consisting of the frames with a range set in **From frame** and **To frame.**

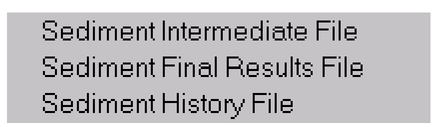
Table 5-1

Controls	Notes	Controls	Notes
1	Current frame		Progress bar
100	Total frame number		The first frame
	Previous frame		Play
	Next frame		The last frame

- **Note**: **According to the tests, the AVI file would NOT be created successfully if the screen resolution is higher than 1280×1024**.

5.3 Visualize Sediments Results

Similarly to flow result files, there are also three kinds of sediment result files.



←(***.mds**)
←(***.sdm**)
←(***.sed**)

Figure 5-8

For sediment intermediate file, you can also choose to load it manually or automatically.

- To load it manually, select **Sediment Intermediate File** in menu Simulation.
- To load it automatically, select **Auto-check Sediment Intermediate Result**. You can set the time interval to check the result.

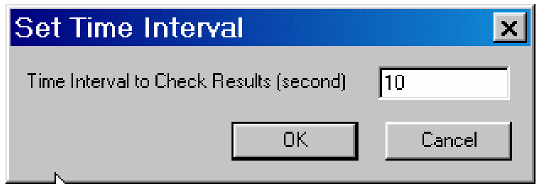


Figure 5-9

For **Sediment Final Results File**, you need to further select the computed sediment result at a certain time.

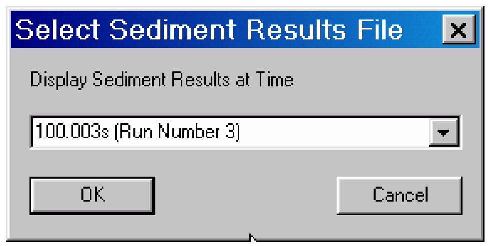


Figure 5-10

After a result file (**mds** or **sdm**) is loaded, you can select sediment variables from **variable selector** on the main toolbar.

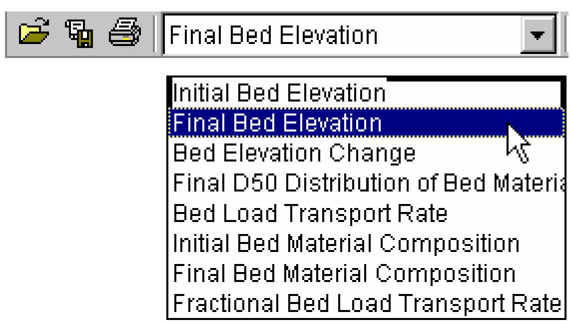


Figure 5-11

If multiple sediment size classes are defined, you can select **Sediment Size Class Display…** in menu **Visualization** to view the results for each size.

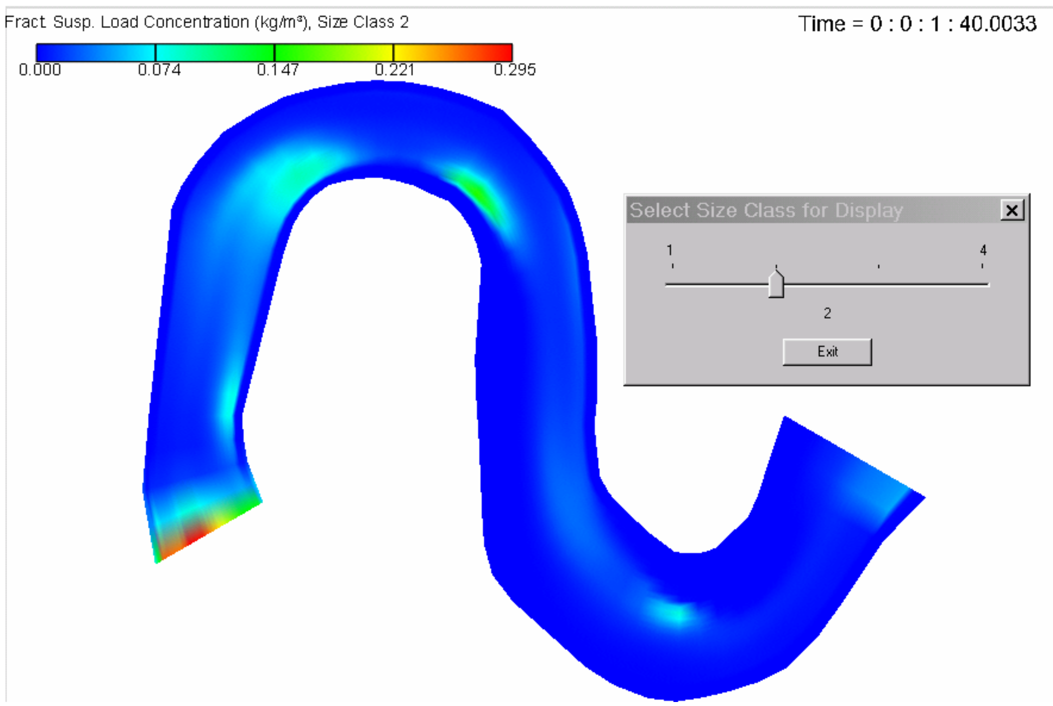


Figure 5-12

If you load **Sediment History File**, you can get help from **History File Editor.** (For details see section 5.2.2)

5.4 Probe and Extract Data

The users can probe and extract data from the simulation results using the graphical tools provided by the GUI.

5.4.1 Probe Data

To probe data, go to menu **Data** and click **Data Probe**… or  on the main toolbar.
There are three kinds of data probes, **Grid Probe**, **Flow Probe** and **Sediment Probe**. All these three data probes use the following two methods to probe data.

- **Probe using I and J indices**:
  - You can input the (I, J) indices of a grid point in **I index** and **J index** and then click **Probe** in **Probe using I and J indices**.

- You can also click one grid point in the domain.
- You can click **Next I**, **Previous I**, **Next J** or **Previous J** to probe the next grid point.

- **Probe using x and y coordinates**:
  - You can input the (x, y) coordinates in **x coordinate** and **y coordinate** and then click **Probe** in **Probe using x and y coordinates**.
  - You can also click one point in the plot area.
  - The grid point closest to the above point (x, y) will be probed.

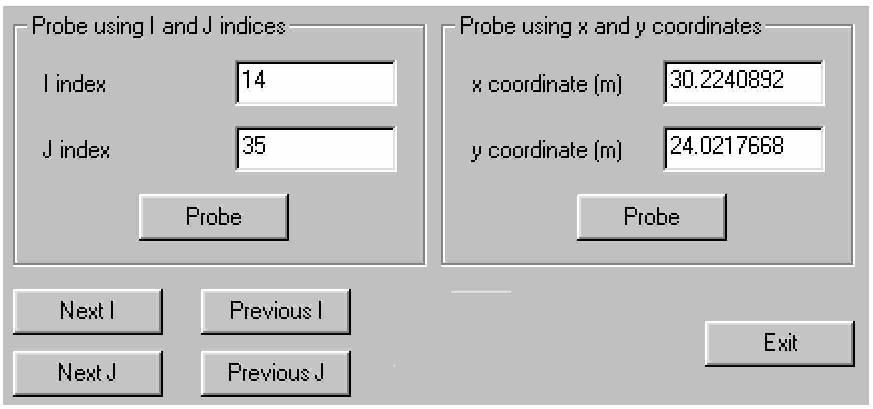


Figure 5-13

For **Grid Probe**, in addition to probe the nodal properties, you can also **Modify properties of the node.**

- You can select grid point either using I and J indices or using x and y coordinates.
- Input the desired values in the **x**, **y**, **z**, **roughness** or **wsl** and then click **Set**.
- You can **Save** you changes, **Undo** one step of previous change and **Undo All** changes.

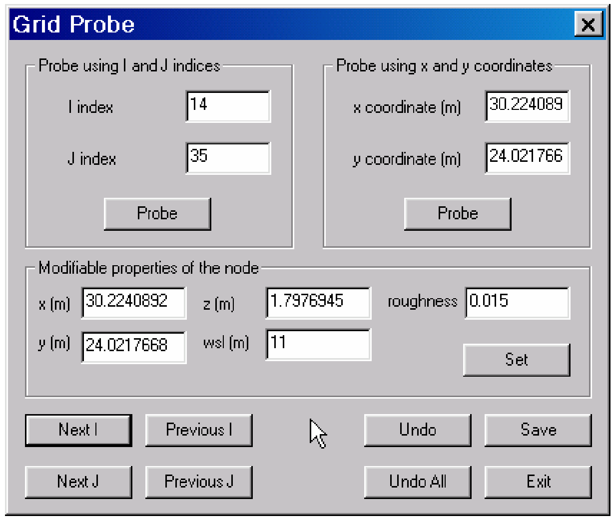
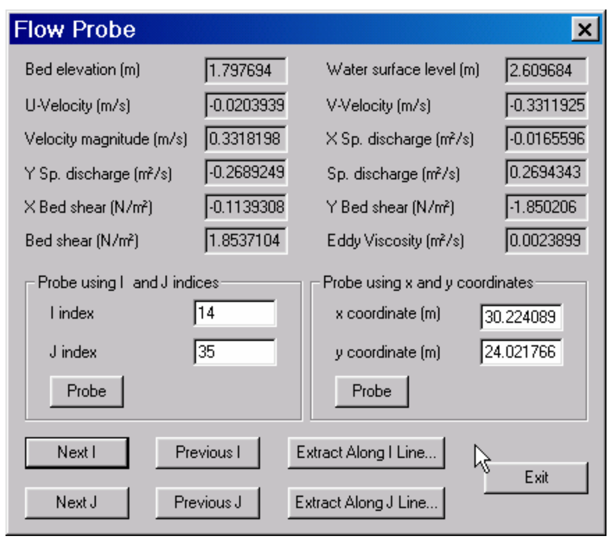


Figure 5-14

For **Flow Probe**, in addition to probe flow data, you can also extract flow data.

- Click **Extract Along I Line…** and the flow data will be extracted to a data file(***.dat**) along a constant I line set in **I index** and highlighted in purple color.
- Click **Extract Along J Line…** and the flow data will be extracted to a data file(***.dat**) along a constant J line set in **J index** and highlighted in purple color.



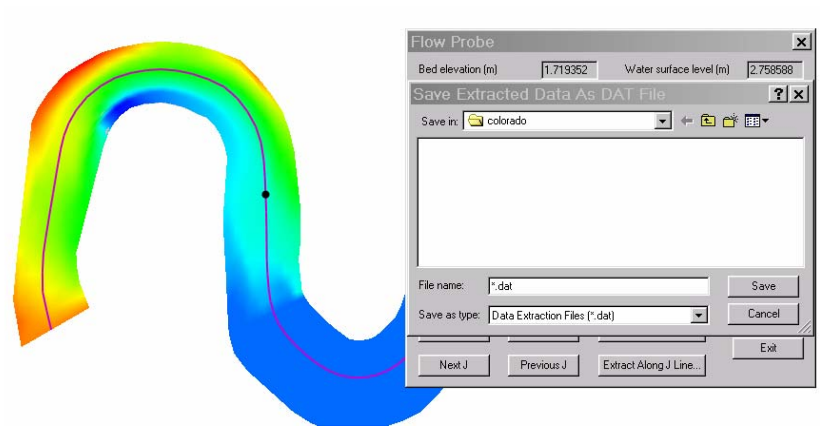
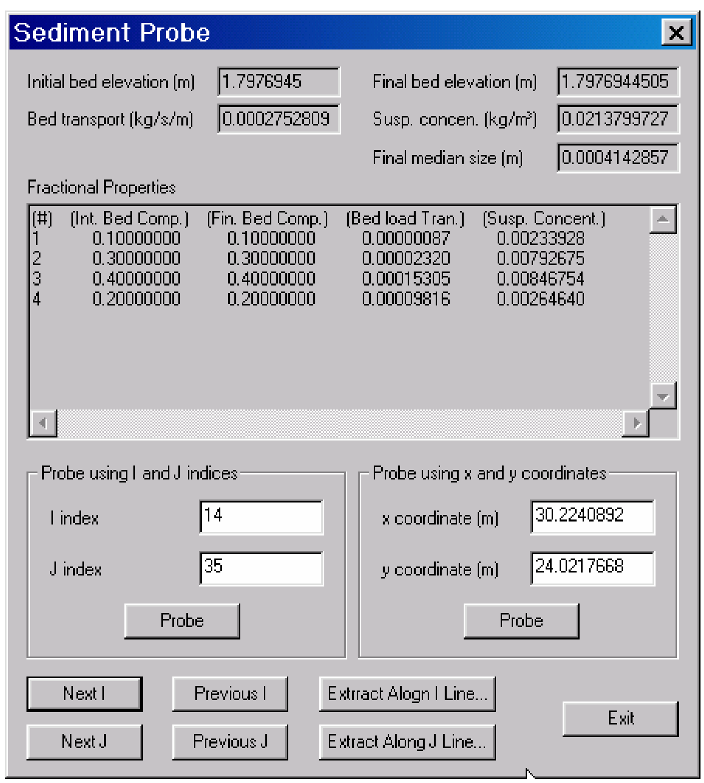


Figure 5-15

Similarly, for **Sediment Probe**, in addition to probe sediment data, you can also extract sediment data.

- Click **Extract Along I Line…** and the sediment data will be extracted to a data file(***.dat**) along a constant I line set in **I index** and highlighted in purple color.
- Click **Extract Along J Line…** and the sediment data will be extracted to a data file(***.dat**) along a constant J line set in **J index** and highlighted in purple color.



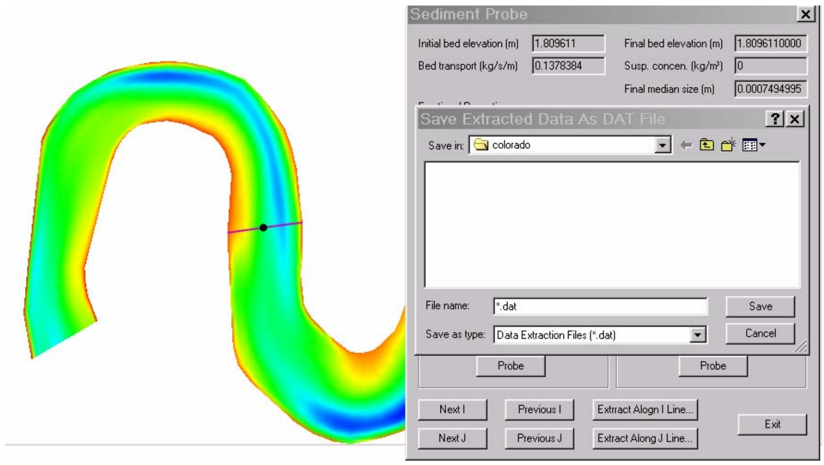


Figure 5-16

5.4.2 Extract Data

Only the **Flow** mode and **Sediment** mode have this capability.

To **Extract Data**, g to menu **Data** and click **Extract Data…**, you need to provide a data extraction file (***.fex**) which contains the (x, y) coordinates of the points you want to extract data. The extracted data will be saved in the same file (***.fex**). The format of file (***.fex**) is the same in **Flow** mode and **Sediment** mode before extraction and different after extraction.

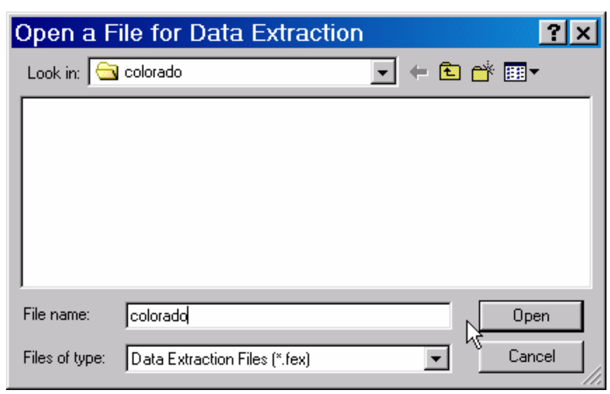


Figure 5-17

To **Extract Data From Polyline**, go to menu **Data** and click **Extract from Polyline…,** the **Polyline for Data Extraction** will appear.

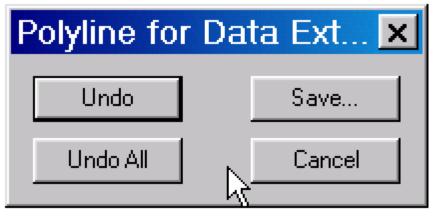


Figure 5-18

- Click the desired place in the domain. For each segment, you need to specify the number of points.

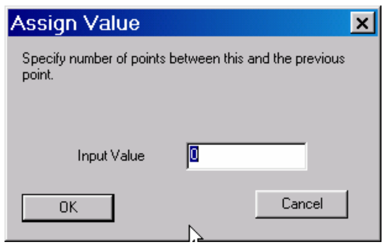


Figure 5-19

- In **Polyline for Data Extraction**,
  - o Click **Save…** to save the extracted data into a file (***.pex**) and the extraction will be finished.
  - o Click **Undo** to delete the last previous point.
  - o Click **Undo All** to delete all the points.
  - o Click **Cancel** to cancel the extraction.

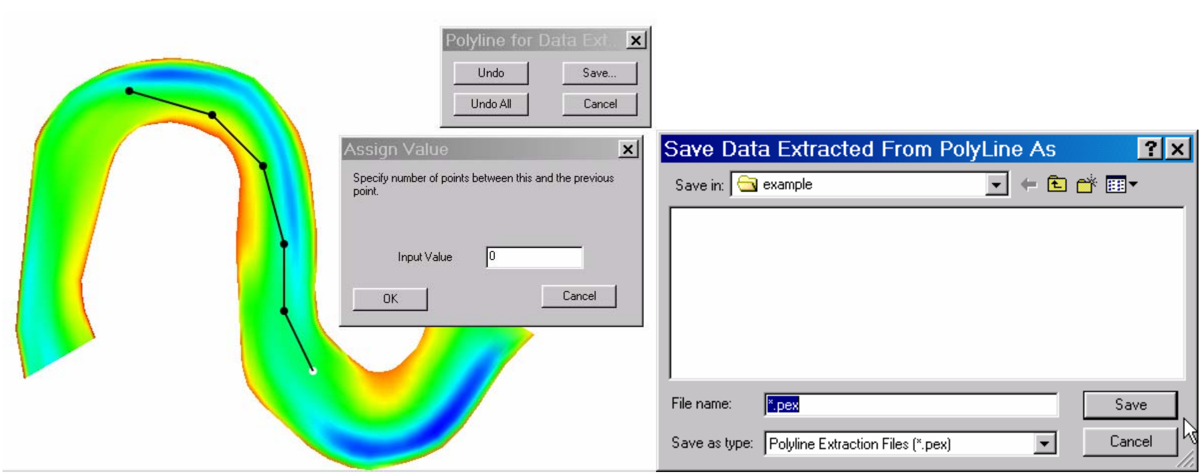
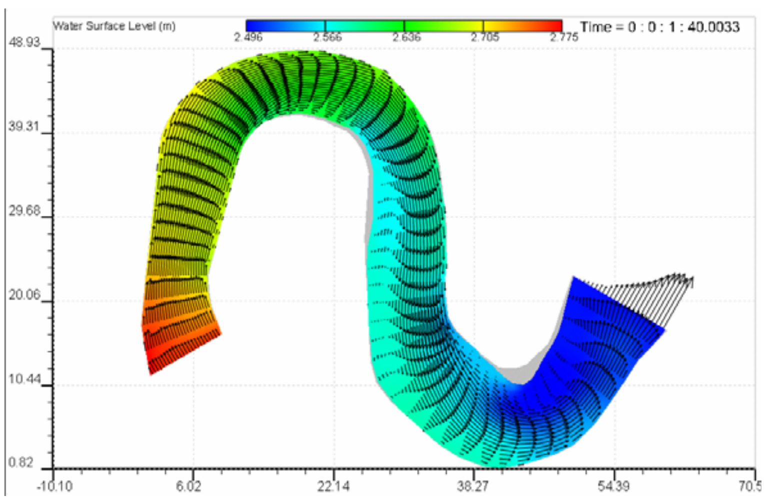


Figure 5-20

5.5 Settings

To personalize your view styles, go to menu **Settings.**

- You can show or hide the **Legend**, **Title**, **Grid**, **Boundary, Simulation Time**, **Dry Area**, **Frame**, **XY Axis** and **Velocity Vector**, and add **Text**.
- You can select **Color Scale**, **Gray Color** or **Reverse Color Map,** set **Background Color** and choose **Plot Types** (four types are available).



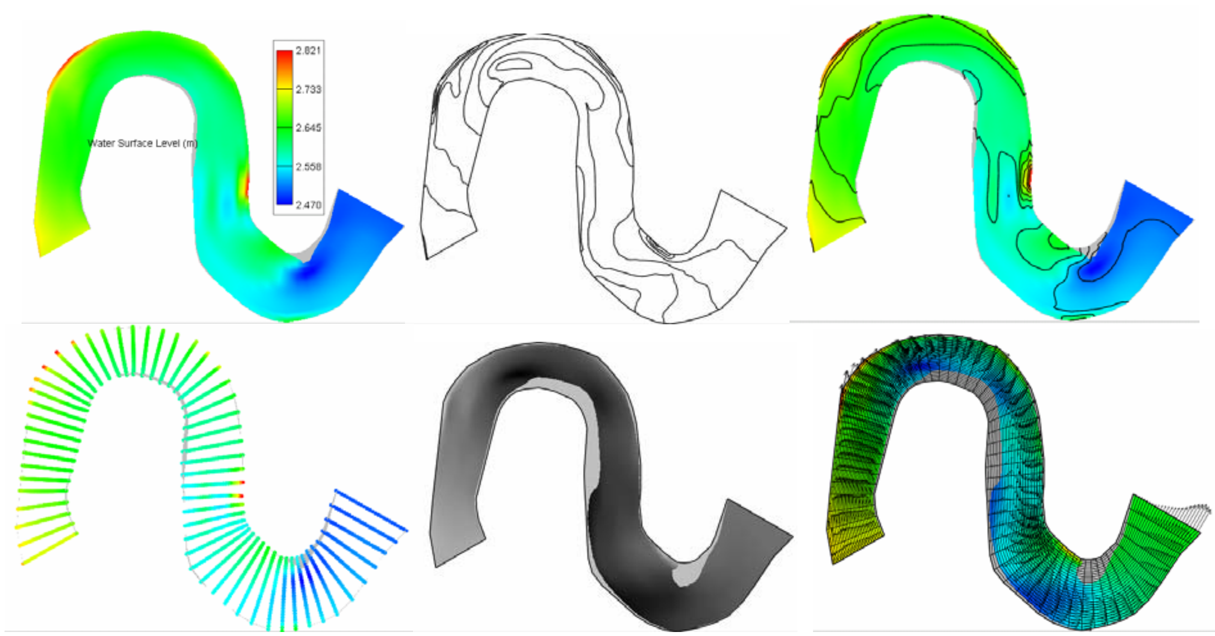


Figure 5-21

5.5.1 Flood Properties

To set flood properties, go to menu **Settings** and click **Flood Properties….**

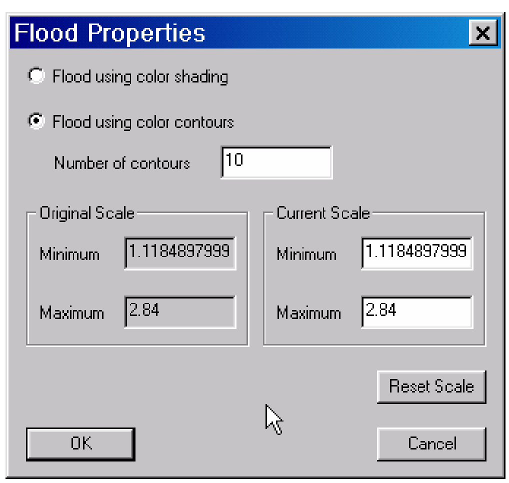


Figure 5-22

In **Flood Properties**,

- Two flood options are available, **Flood using color shading** and **Flood using color contours**. If you select the second option, you can specify the **Number of contours**.
- For scales of color bar, you can edit the color bar by setting the new **Minimum** and **Maximum** values and you can restore the **Original Scale** by clicking **Reset Scale.**
- To reverse color map, select **Reverse Color Map** in menu **Settings**.

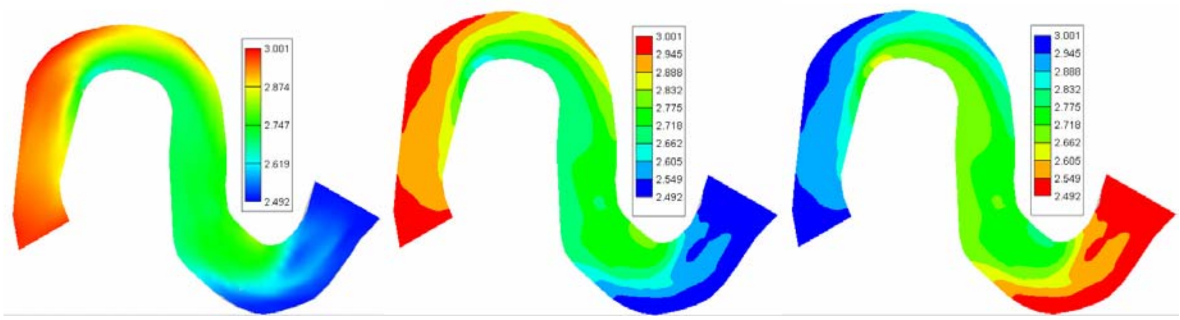
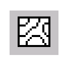
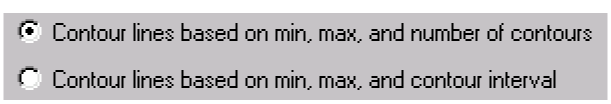


Figure 5-23

5.5.2 Contour Properties

To set Contour Properties, first set **Plot Type** as **Lines**  or **Lines and Flood** , and then select **Contour Properties…** or **Contour Labels Properties…**in menu **Settings.**

In **Contour Lines Properties**, you can set plot options for contour lines. Two options are available. For each option, you need to set the **Number of contours** and **Contour interval** correspondingly.



For scales of color bar, you can set the new **Minimum** and **Maximum** values and **Reset Scale** to the **Original Scale.** You can also **Set Contours Color…**

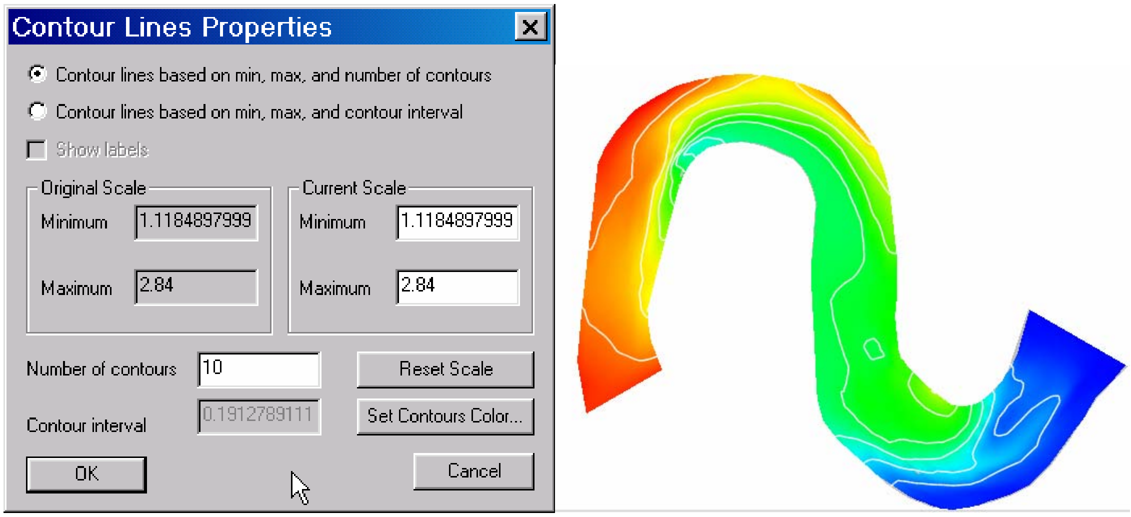


Figure 5-24

In **Contour Labels**, first select the contour line in **Contour Selector** and the current contour will be displayed in **Contour Value.**

- To add label, click **Add Label** and click the desired place on the corresponding contour highlighted as purple color. You can add multiple labels for one contour line.
- To set the rotation angle for current label, input the desired angle in **Rotation Angle.** This angle will be applied to the currently selected contour value.
- To remove label, click **Remove Label** and then click the label you want to delete.
- To remove all labels of one contour line, click **Remove All Labels of One Line.**
- To remove all labels of all contours, click **Delete All Labels.**
- To set font size and significant digit for all labels, input the desired values in **Font scaling factor** and **Significant digits** and then click **Apply** in **Label Properties (all labels).**
- To set font color for all labels, Click **Font Color...**
- To set background for all labels, check or uncheck the **Font background** to make the label show the background color or transparent.
- **Note:** Please make sure to add labels onto the correct contour line when adding labels.

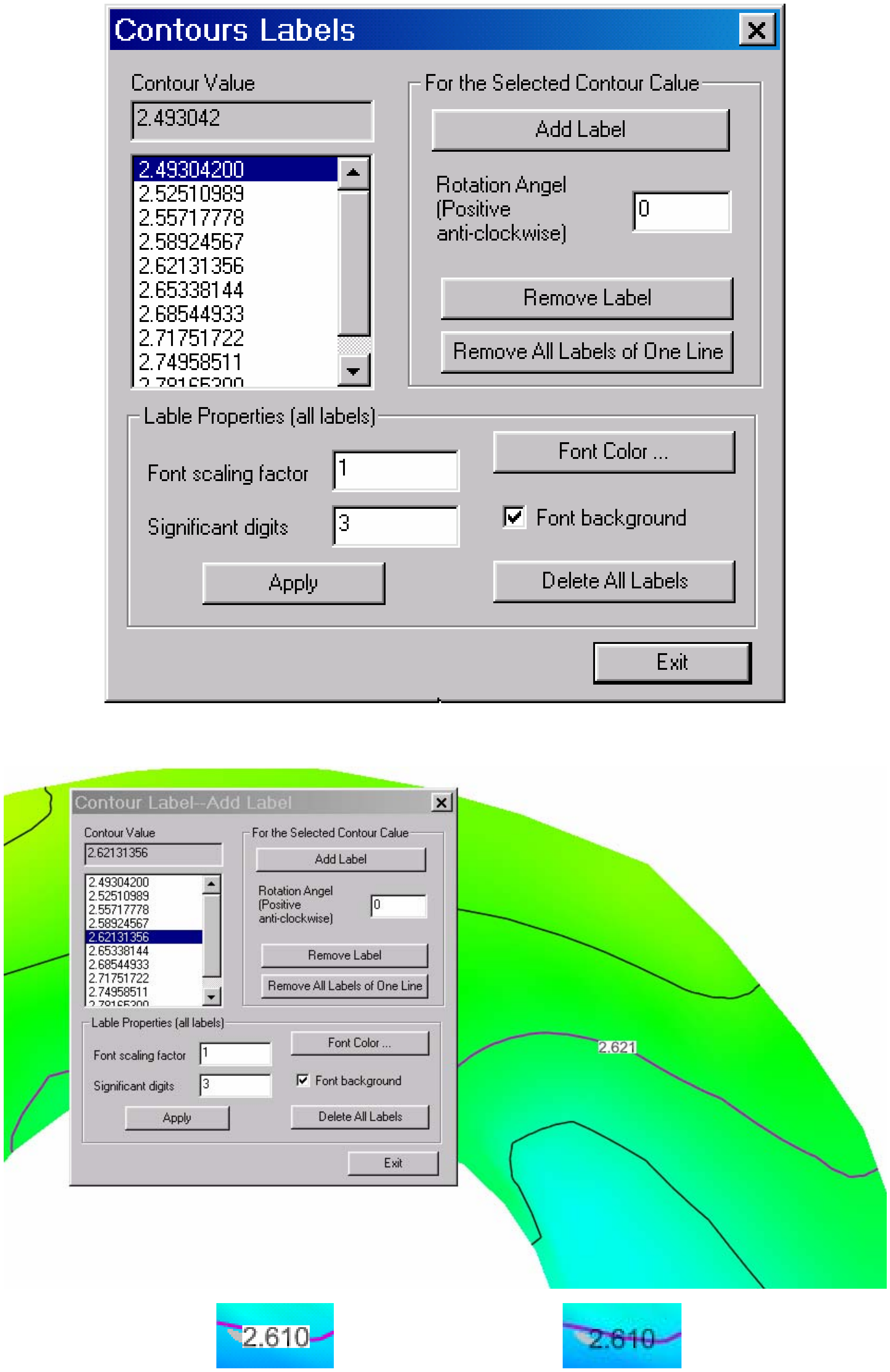

Contours Labels
Contour Value
2.493042
2.49304200
2.52510989
2.55717778
2.58924567
2.62131356
2.65338144
2.68544933
2.71751722
2.74958511
For the Selected Contour Calue
Add Label
Rotation Angel (Positive anti-clockwise)
0
Remove Label
Remove All Labels of One Line
Lable Properties (all labels)
Font scaling factor
1
Significant digits
3
Font Color ...
Font background
Apply
Delete All Labels
Exit
Contour Label--Add Label
2.62131356
2.621
2.610
2.610

Figure 5-25

5.5.3 Vector Properties

To set properties of velocity vector, click **Vector Properties…** in menu **Settings**. In **Vector Parameters,**

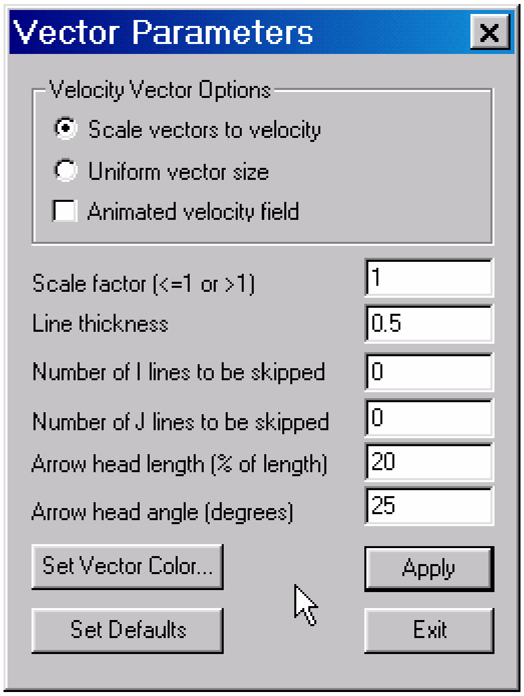


Figure 5-5

- **Set Vector Scale Options**: You can choose to **Scale vectors to velocity** or set the **Uniform vector size**. If you check the **Animated velocity field**, an animation effect will be shown on the velocity field.
- **Set Display Parameters**: You can choose not to display the whole velocity field by specifying **the Number of I lines to be skipped** and the **Number of J lines to be skipped.**
- **Set Vector Parameters**: You can set the **Scale factor**, **Line Thickness**, **Arrow head length**, **Arrow head angle.**
- **Set Vector Color**, click **Set Vector Color…**
- **Load Default Values:** Click **Set Defaults.**

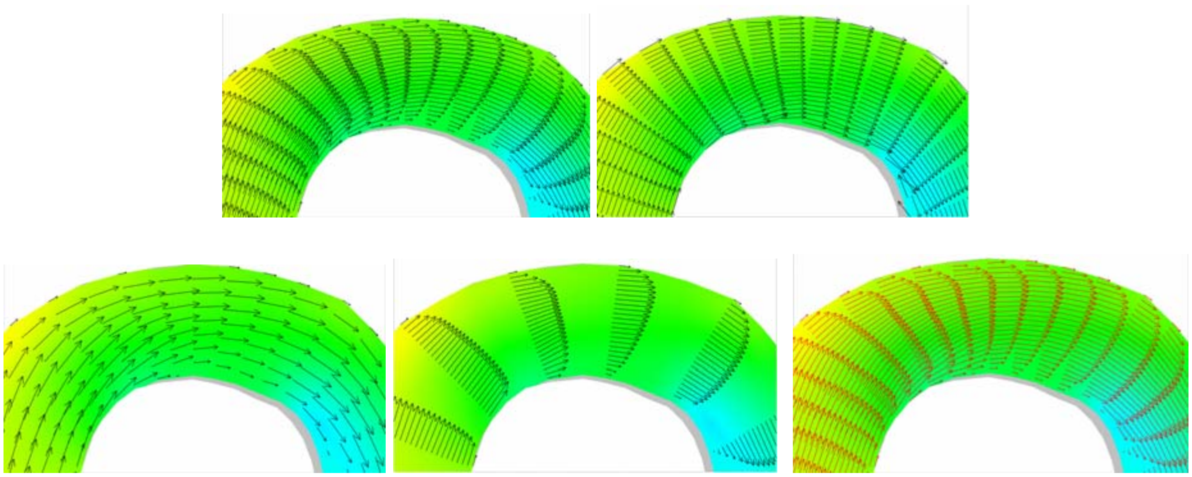


Figure 5-6

5.5.4 Text

There are two statuses for texts, adding and editing. In adding status, the editing functions, such as **Replace Text**, **Delete Text**, and **Delete All**, will be disabled. They will be enabled in editing status.

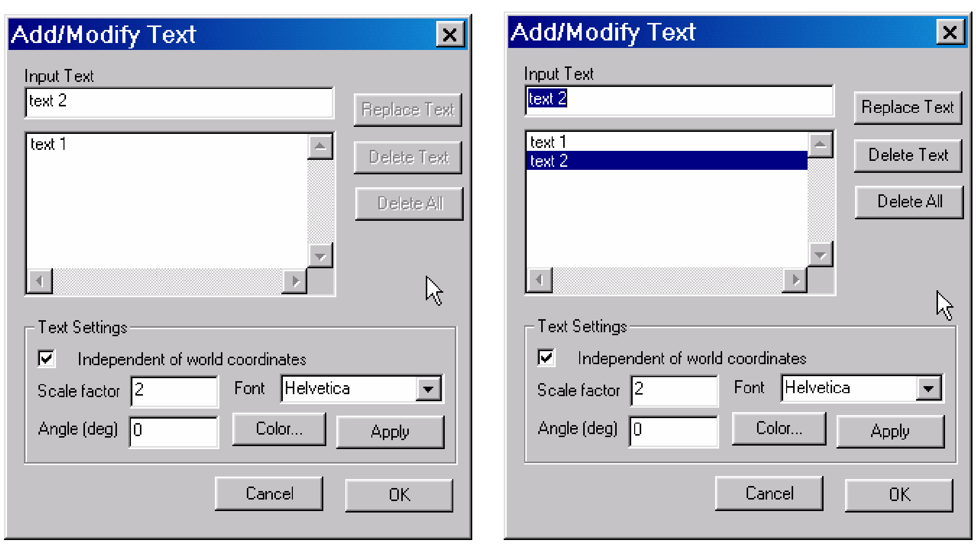
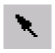


Figure 5-28

To **Add New Text**,

- Click  or **Add Text…** in menu **Settings** and then click the desired place in the plot area.
- In **Add/Modify Text**, input the desired text in **Input Text** and then click **OK.**

To **Edit Texts**,

- Click  and then **Double-click** the text you want to edit. The current text will be highlighted and displayed in **Input Text.**
- In **Add/Modify Text**,
    - **To Replace Text**: Select the text in **Text Selector** and input the new text in **Input Text**, then click **Replace Text**.
    - **To Delete Text**: Select the text in **Text Selector** and then click **Delete Text**.
    - **To Delete All Texts**: Click **Delete All**.
    - You can set the current text coordinates **dependent** or **independent of world coordinates** to make it attached or detached to the real coordinates system of the mesh.
    - **To Set Font Size for Current Text**: Input the desired **Scale factor** and then click **Apply**.
    - **To Set Font Angle for Current Text**: Input the desired **Angle** and then click **Apply**.
    - **To Set Font Color for Current Text**: Click **Color...**
    - **To Set Font for All Texts**: Select the font in **Font** and then click **Apply**.
    - **To Move Text**: Click  and then click the text you want to move and hold it. Then drag it to the desired place and release the mouse.

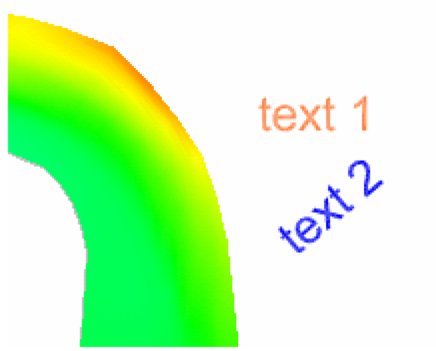
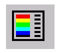
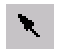
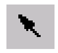


Figure 5-29

5.5.5 Legend

To show/hide Legend, click  or **Show Legend** in menu **Settings**.

To move legend, click  and then click the legend and hold it. Then drag it to the desired place and release the mouse.

To set legend properties, click  and **Double-click** the legend.

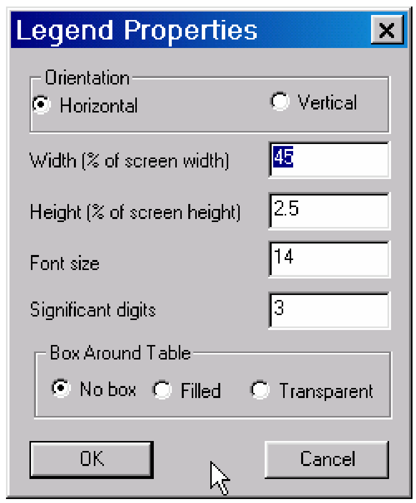
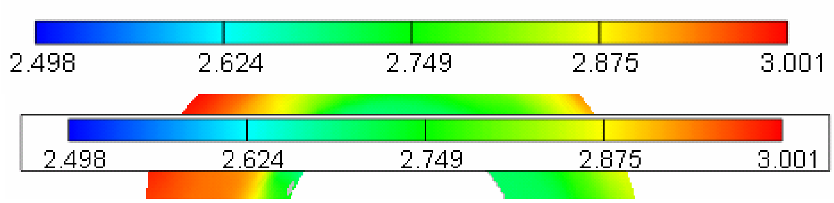


Figure 5-30

In **Legend Properties**

- You can set the **Horizontal** or **Vertical Orientation** of the legend.
- You can set the **Width** and **Height** of the legend with respect to the screen width and screen height.
- You can set the **Font size** and **Significant digit**.
- You can set the property of **Box Around Table**, **No Box**, **Filled** or **Transparent**.
- Click **OK** when all properties have been set.



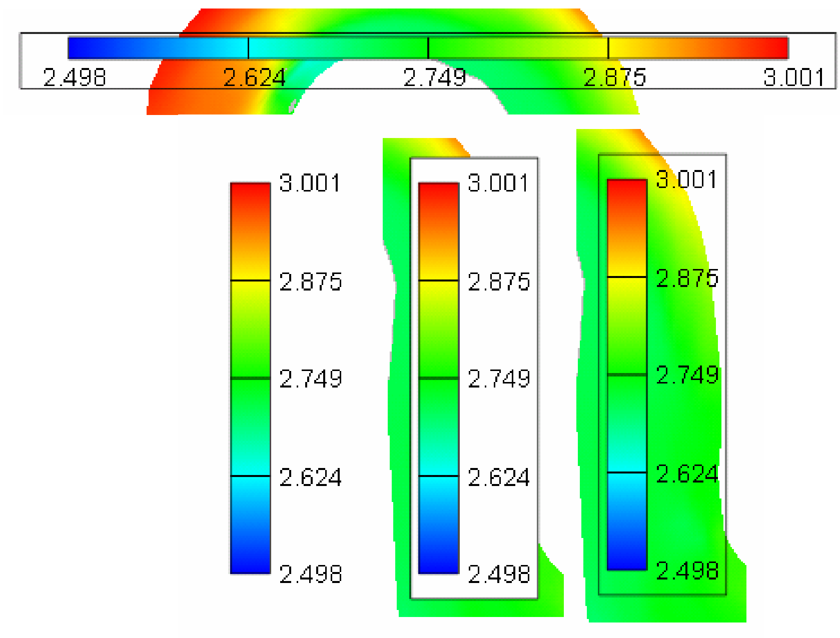


Figure 5-31

5.5.6 Background

You set background color and background image.

To set background color, click **Background Color…**in menu **Settings** and select the background color.

To set background image,

- **Import Background Image:** Click **Import Bitmap Image…** in menu **File.**

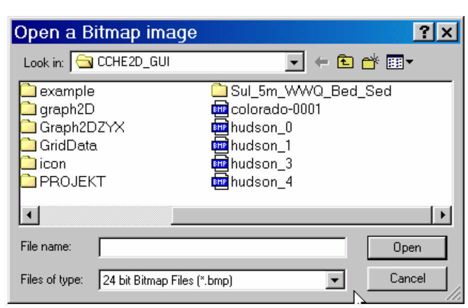
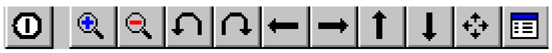


Figure 5-32

- **Activate Transformation Toolbar:** Click or check **Edit Background Image** in menu **Settings** to start editing the image.



- **Set Transformation Parameters:** Click . In **Transformation Parameters**, you can set the **Scaling Step**, **Rotation Step** and **Translation Step.**

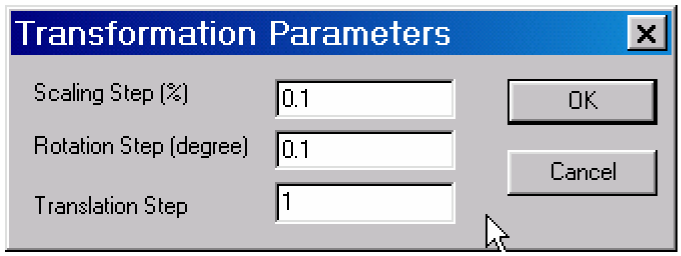


Figure 5-33

- **Set Appropriate Scale:** Click or to zoom in or zoom out the background image incrementally at a **Scaling Step** set in **Transformation Parameters**.
- **Set Appropriate Angle:** Use to rotate the background image clockwise or anticlockwise.
- **Set Appropriate Translation:** Use to move the background image leftward, rightward, upward or downward; Use to move background image toward any direction.
- **Save Transformation Settings**: Click or uncheck **Background Image** in menu **Edit** to stop editing the image. The transformation settings will be saved as a geometrically referenced file (***.grm**) in the same directory where the background image is located. Next time when importing this image, the background image will be transformed according to the settings in that file.

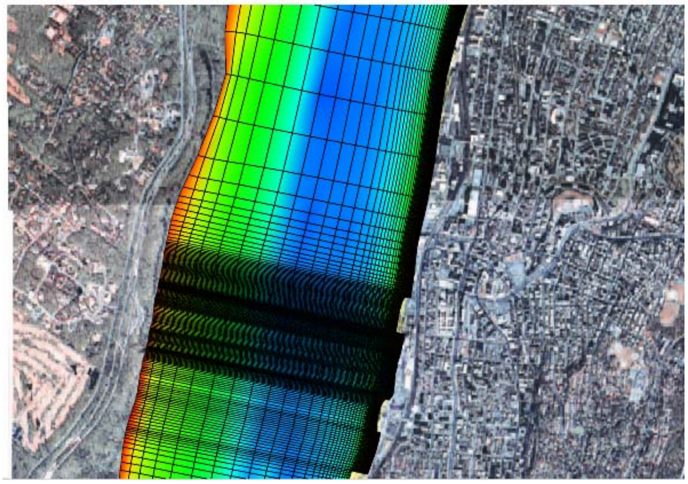


Figure 5-34

5.5.7 Layout

You can save the current visualization settings as a layout file (***.lof**).

- To Save Layout File: Click **Save Layout…**in menu **File**.
- To Open Layout File: Click **Open Layout…**in menu **File**. You need to set the **File Association Options** for Layout File. If you want to Associate a new file with the layout, you need to provide the path of the new geo file.
-

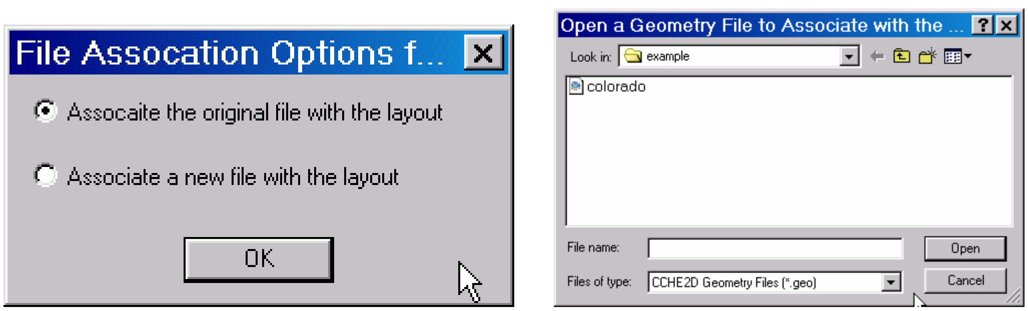


Figure 5-35

6 APPENDIX: FILE FORMAT

File Name	File Format Description (Pseudo-Fortran Code)	File Name	File Format Description (Pseudo-Fortran Code)
Mesh File (*.geo)	Imax, Jmax Do j = 1, Jmax Do i = 1, Imax x (x or east coordinate), y (y or north coordinate), wsl (initial water surface), bed (initial bed elevation), lsl (nodal type), roughness Enddo Enddo	**Monitor Points File (*.mon)**	Number of Monitor Points (<= 5) Do i = 1, Number of Monitor Points I index, J index Enddo
Discharge Hydrograph File (*.dhg)	Number of Ordinates Do i =1, Number of Ordinates Time (second), Discharge (m^3/s) Enddo	**Rating Curve File (*.rcv)**	Number of Ordinates Do i =1, Number of Ordinates Stage (m), Discharge (m^3/s) Enddo
Stage Hydrograph Curve Files (*.shg)	Number of Ordinates Do i =1, Number of Ordinates Time (second), Stage(m) Enddo	**Data Extract File Before Extraction (*.fex)**	Number of Points Do i = 1, Number of Points x, y Enddo

References

Jia, Y.F. and Wang, Sam S.Y. (2001). "CCHE2D: Two-dimensional Hydrodynamic and Sediment Transport Model for Unsteady Open Channel Flow Over Loose Bed." NCCHE Technical Report. NCCHE-TR-2001-01, Aug. 2001

Wu, W.M. (2001). "CCHE2D Sediment Transport Model version 2.1." NCCHE Technical Report. NCCHE-TR-2001-03, Aug. 2001

Khan, A.A. (2002). "CCHE2D-GUI: Graphical User Interface for CCHE2D Model. Users Manual—Version 2.0." NCCHE Technical Report. NCCHE-TR-2002-04, 2002.

Zhang, Yaoxin (2005). "CCHE2D Mesh Generator Users' Manual—Version 2.50." NCCHE Technical Report. NCCHE-TR-2005-01, Jan 2005.

Zhang, Yaoxin (2005). "CCHE2D-GUI Quick Start Guide." NCCHE Technical Report. NCCHE-TR-2005-04, June 2005.