

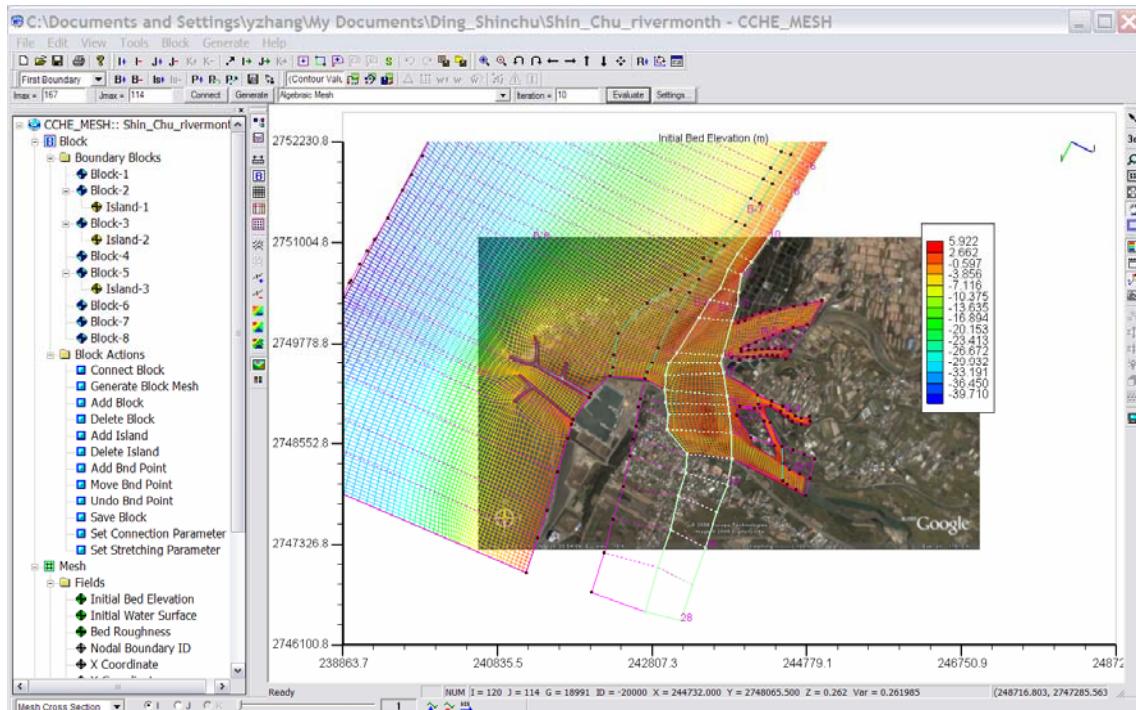
NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

CCHE-MESH: 2D Structured Mesh Generator User's Manual -Version 3.x

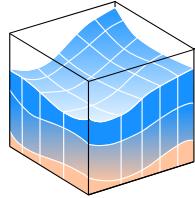
Technical Report No. NCCHE-TR-2009-01

Feb 2009

Yaoxin Zhang and Yafei Jia



School of Engineering
The University of Mississippi
University, MS 38677



NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

Technical Report No. NCCHE-TR-2009-01

CCHE-Mesh: 2D Structured Generator
User's Manual – Version 3.x

Yaoxin Zhang
Research Scientist

Yafei Jia
Research Professor

The University of Mississippi

Feb 2009

Table of Contents

1 INTRODUCTION.....	1
1.1 CCHE-MESH.....	1
1.1.1 METHODOLOGIES AND TECHNIQUES.....	1
1.2 USING THIS MANUAL.....	1
1.3 WHAT'S NEW.....	2
2 MESH GENERATION	3
2.1 INTRODUCTION	3
2.2 ALGEBRAIC MESH GENERATION.....	4
2.2.1 TWO-BOUNDARY METHOD	4
2.2.2 STRETCHING FUNCTION.....	5
2.3 NUMERICAL MESH GENERATION	7
2.3.1 TTM MESH GENERATION SYSTEM.....	7
2.3.2 RL MESH GENERATION SYSTEM.....	8
2.3.3 IMPROVED TTM AND RL SYSTEMS WITH SMOOTHNESS FUNCTION.....	9
2.3.4 IMPROVED RL SYSTEMS WITH CONTRIBUTION FACTORS	10
2.3.5 IMPROVED RL SYSTEMS WITH CONTROLS ON DISTORTION FUNCTION.....	13
2.3.6 IMPROVED RL ADAPTIVE MESH GENERATION SYSTEM	13
2.4 MESH EVALUATION	14
3 BED INTERPOLATION	16
3.1 INTRODUCTION	16
3.2 INTERPOLATION METHODS	16
3.2.1 INVERSE DISTANCE WEIGHTING METHOD	16
3.2.2 PLANAR INTERPOLATION METHOD	17
3.3 INTERPOLATION ALGORITHMS	18
3.3.1 RANDOM DATABASE	18
3.3.2 STRUCTURED DATABASE	19
4 GRAPHICAL USERS INTERFACE.....	21
4.1 USERS INTERFACE.....	21
4.1.1 MAIN WINDOW.....	21

4.1.2 TOOLBARS	22
4.1.2.1 Standard Toolbar.....	22
4.1.2.2 Mesh Generation Toolbar	23
4.1.2.3 Mesh Editing Toolbar	23
4.1.2.4 Block Editing Toolbar.....	24
4.1.2.5 Tools Toolbar.....	25
4.1.2.6 Image Editing Toolbar	25
4.1.2.7 View Toolbar	26
4.1.2.8 View Tool Toolbar.....	26
4.1.3 MENUS	27
4.1.3.1 File Menu	27
4.1.3.2 Edit Menu	28
4.1.3.3 View Menu	29
4.1.3.4 Tools Menu	30
4.1.3.5 Block Menu.....	31
4.1.3.6 Generate Menu.....	32
4.1.3.7 Help.....	32
4.2 FILE MANAGEMENT	33
4.3 DEFINE BLOCKS	36
4.3.1 IMPORT A TOPOGRAPHY IMAGE	36
4.3.1.1 Load Topography Image.....	36
4.3.1.2 Coordinates Transformation	38
4.3.1.3 Digitize Image.....	40
4.3.2 IMPORT A TOPOGRAPHY DATABASE	42
4.3.3 IMPORT MEASURED CROSS SECTIONS.....	44
4.3.3.1 Load Measured Cross Sections	45
4.3.3.2 Refine Measured Cross Sections.....	46
4.3.4 IMPORT DIGITAL ELEVATION MODEL	50
4.3.5 IMPORT SHAPE FILE.....	53
4.3.6 DEFINE BLOCK BOUNDARY.....	55
4.3.7 SET BLOCK CONNECTION PARAMETER.....	61
4.3.8 EDIT BLOCK BOUNDARY	62
4.3.9 LIMITATIONS OF MULTI-BLOCK DEFINITIONS.....	66
4.4 GENERATE MESH	68
4.4.1 GENERATE ALGEBRAIC MESH.....	68
4.4.2 GENERATE QUALITY MESH.....	77
4.4.3 GENERATE ADAPTIVE MESH	83
4.5 EVALUATE MESH	85
4.6 BED ELEVATION INTERPOLATION	86
4.6.1 BED INTERPOLATION	87
4.6.2 BED SMOOTHING	91
4.7 EDIT MESH.....	92
4.7.1 EDIT MESH.....	93
4.7.2 EDIT FIELD AND NODAL ID	99

Table of Contents	iii
-------------------	-----

4.8 VISUALIZATION.....	103
4.8.1 2D XY PLOT	103
4.8.2 VIEW TOOLS	110
4.8.3 SHAPE FILES	127
4.8.4 DISPLAY OPTIONS.....	130
4.8.5 BACKGROUND	135
4.9 FILE FORMATS	138
4.9.1 TOPOGRAPHY DATABASE FILE (*.MESH_XYZ).....	139
4.9.2 MEASURED CROSS SECTIONS FILE (*.MESH_MCS)	139
4.9.3 MESH GEO FILE (*.GEO)	140

1 INTRODUCTION

1.1 CCHE-MESH

Welcome to use CCHE-MESH version 3.0! The CCHE-MESH is a 2D structured mesh generator with professional and friendly graphical users interface. It allows rapid quality mesh generation from topography map and database. And, it provides the users the file input and output (I/O) management, the algebraic mesh generation, the numerical mesh generation, the mesh editing, and the operation on topography database.

1.1.1 Methodologies and Techniques

The following methodologies and techniques have been integrated into the CCHE-MESH. The technical details can be found in Chapter 2.

- A two-boundary algebraic mesh generator;
- A three-parameter EDS stretching function;
- A multi-block algorithm for complex geometries;
- An improved RL mesh generator with smoothness functions;
- An improved RL mesh generator with contribution factors;
- An improved RL adaptive mesh generator;
- An improved TTM mesh generator with smoothness functions;
- A Laplacian smoothing method;

1.2 Using This Manual

This manual provides the users necessary information to use the CCHE-MESH. It explains in detail the mesh generation methodologies and techniques, how to create boundary information from scratch, how to generate mesh using different generation systems, how to

interpolate the bed elevation, how to edit the nodal properties, and how to view the resulting mesh. It is organized as follows:

- Chapter 2 gives a detailed technical description on the mesh generation methodologies and techniques integrated in the CCHE-MESH. The users, especially those new to the mesh generation, are strongly recommended to read this chapter carefully before using the CCHE-MESH.
- Chapter 3 introduces the bed interpolation methods and algorithms. The users will learn how to deal with different types of database to get best interpolation results.
- Chapter 4 describes in detail the capabilities and the how-to's of the CCHE-MESH. The users will learn how to generate quality meshes from scratch using the CCHE-MESH.

1.3 What's New

The CCHE-MESH version 3.0 has the following main new features:

- A newly designed graphical users' interface.
- A newly designed and more robust core data structure.
- A newly developed adaptive mesh generator.
- Improved bed interpolation algorithms.
- Improved algebraic mesh generator.

2 MESH GENERATION

2.1 Introduction

Computational fluid dynamics (CFD) is based on solving a set of highly non-linear partial differential equations (P.D.E) on a physical domain, which is usually discretized and represented by a computational mesh. Despite the numerical method used, the success of solving these P.D.Es depends largely on the mesh quality. As the general academic criteria, the orthogonality and smoothness are often used to evaluate the mesh quality quantitatively. The adaptivity, referring to the control of the mesh density distribution according to the physics of a particular problem, is often required to evaluate the mesh quality.

Basically, there are two types of meshes used in CFD: the structured and the unstructured. The structured meshes consist of families of mesh lines with the property that members of a single family do not cross each other and cross each member of the other families only once, while the unstructured mesh does not have such a restriction. The advantage of the structured mesh is that any mesh node is uniquely identified by a set of two (2D mesh) or three indices (3D mesh) and thus is easy to access. In the unstructured meshes, a connection table is required to identify the relationship of the mesh nodes. Usually, the structured mesh is used for the Finite Difference Method (FDM) and the Finite Volume Method (FVM), while the Finite Element Method (FEM) often uses the unstructured mesh. The CCHE-MESH Mesh Generator is developed to generate the structured meshes.

The methods applied in the structured mesh generation are grouped into two categories: the algebraic methods and the numerical methods. The numerical methods solve a set of P.D.Es to determine the mesh distribution while the algebraic methods generate mesh directly by interpolation. The numerical methods are global approaches and can provide meshes with smooth transitions and orthogonality maintained. Although the algebraic method can always resolve a mesh with the minimum computational effort, the quality of the mesh, such as smoothness and orthogonality, is often not globally satisfactory, especially when the computational domain is complex. As a result, smoothing or optimizing methods have to be used to further improve the mesh. This is the main disadvantage of the algebraic methods.

Nevertheless this method is often used to provide the initial mesh for the numerical mesh generation. For the domains with complex geometry, the multi-block scheme is often adopted.

2.2 Algebraic Mesh Generation

Fast computation and direct control of mesh nodes are the two main advantages of the algebraic mesh generation which interpolates the interior mesh nodes directly from the boundaries.

2.2.1 Two-boundary Method

In the CCHE-MESH, a two-boundary method combined with a multi-block scheme is used to generate the algebraic meshes. For demonstration purpose, a simple single-block domain is used, as shown in Figure 2-1. As can be seen, a domain consists of two kinds of boundaries: the outer boundaries and the inner boundaries. The shape of the domain is controlled by the outer boundaries, while the area surrounded by the inner boundaries is considered to be “outside” of the domain. That is, the mesh nodes in this area are inactive during the numerical simulation. Both the outer boundaries and the inner boundaries are composed of the top boundary, bottom boundary, left boundary, and right boundary. In the two-boundary method, only the top boundary and the bottom boundary are independent and used to control the geometry, while the left boundary and the right boundary are dependent on the top and bottom boundaries. The equal number of control points is distributed along the top and bottom boundaries to approximate the boundary curves. The control points divide the whole domain into sub-sections.

Basically three steps are involved in the two-boundary algebraic mesh generation.

- Define the outer boundaries and the inner boundaries by placing the boundary control points.
- Distribute the equal number of boundary points along the top and the bottom boundaries. Each pair of the boundary points forms a control line.
- Distribute the internal mesh nodes along the control lines.

The nodal distribution can be controlled by the stretching function.

Note: In later sections and chapters of this manual, the “Bottom” boundary and the “Top” boundary will be referenced as the “First” boundary and the “Second” boundary.

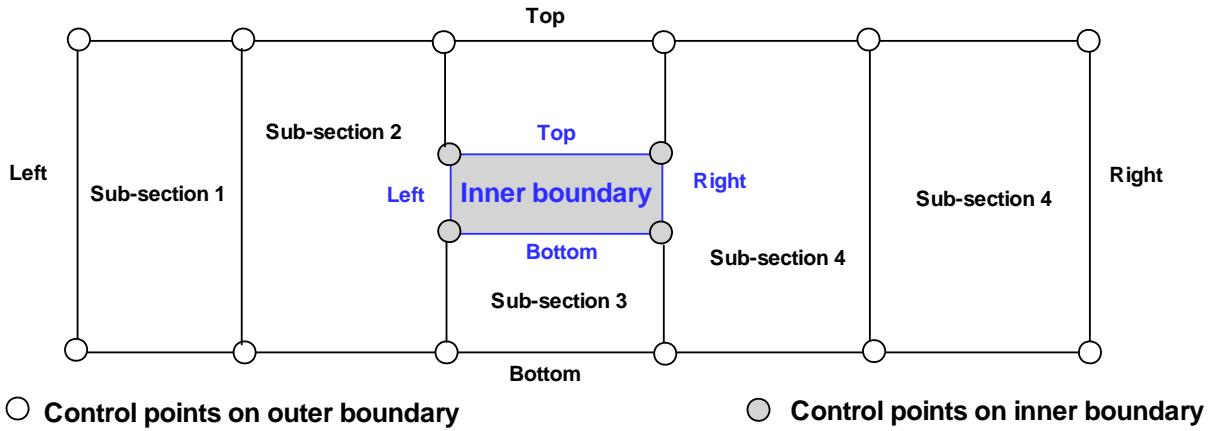


Figure 2-1 A single-block domain

2.2.2 Stretching Function

The stretching function is widely used in the algebraic mesh generation. In CCHE-MESH, a more flexible and powerful two-direction stretching function EDS is proposed.

$$s_j = \sum_{i=1}^{j-1} \left[\frac{2}{\exp(\phi) + \exp(-\phi)} \right]^E / \sum_{i=1}^{N-1} \left[\frac{2}{\exp(\phi) + \exp(-\phi)} \right]^E, \quad (2.1a)$$

$$\phi = \left[\frac{j-1}{N} - D \right] \times S, \quad (2.1b)$$

where s_j is the relative location; j is the label of one point; N is the total number of points along a mesh line; E ($= -1, 0, 1$) is the exponential parameter; D ($0 \leq D \leq 1$) is the deviation parameter; S (>0) is the parameter used to control the degree of stretching, called scale parameter.

With this stretching function, the location of any node in one line AB (Figure 2-2) is calculated by

$$x_j = x_A + (x_B - x_A) \cdot s_j \quad (2.2)$$

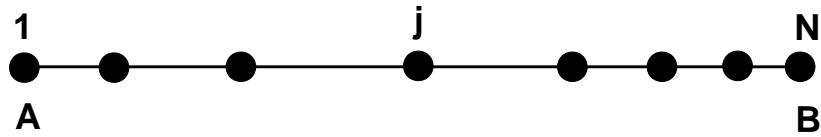


Figure 2-2 Nodal distributions on one line

Figure 2-3 illustrates the effects of these three parameters, E, D, and S. The exponential parameter determines the characteristic of the distribution: contraction to a point, repulsion from a point, or uniformity. If $E = -1$, the distribution is contracting to the point; if $E = 1$, the distribution is repulsing from the point; and if $E = 0$, the distribution is uniform. The deviation parameter provides the relative location of this point along AB. For example, if $D = 0.5$, this point is located at the center. The scale parameter S controls the degree of stretching. The larger S is, the more the distribution is stretched. If $S = 0$, the distribution is uniform. Note that in this EDS stretching function, the three parameters can be of any values. The reference values provided here would make it easy for using the function.

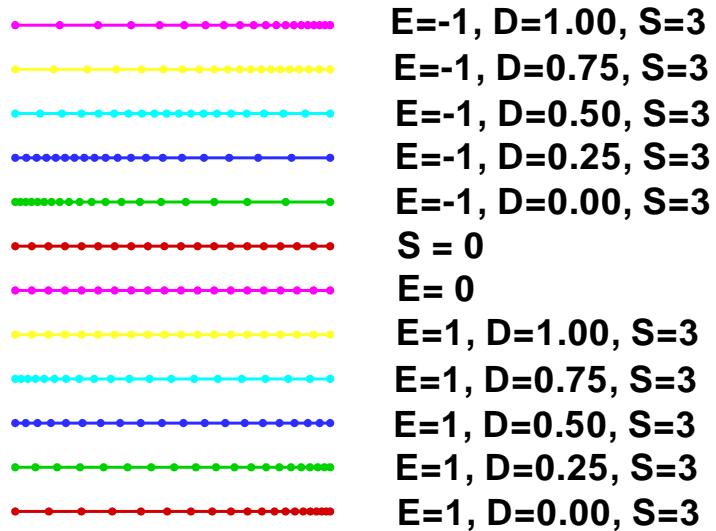


Figure 2-3 Effects of E, D, and S

2.3 Numerical Mesh Generation

Compared to the algebraic mesh generation, the numerical mesh generation is more sophisticated and thus can produce more reliable meshes with higher quality. The disadvantages lie in two aspects: (1) it requires more computation efforts, and (2) it is more difficult to use. In the numerical mesh generation, a set of P.D.Es is solved to obtain the desired mapping between the physical domain and the computational domain. The P.D.E systems used for the numerical mesh generation include the elliptic systems, the hyperbolic systems, the parabolic systems and the variational Laplacian systems.

2.3.1 TTM Mesh Generation System

The elliptic system is the most widely used in 2D mesh generation, which is derived from the Laplace equation for the stream function and the velocity potential function. Two elliptic systems are broadly used: the TTM (Thompson et al., 1977) and the RL (Ryskin and Leal, 1983).

The elliptic system used by Thompson et al. (1977) for the numerical mesh generation is the Poisson equation with the following forms:

$$\frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} = P(\xi, \eta) \quad (2.3a)$$

$$\frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = Q(\xi, \eta) \quad (2.3b)$$

where (ξ, η) is the computational coordinate system and (x, y) is the physical coordinate system; and P, Q are control functions.

If P and Q are zero, Equation (2.3) is reduced to the Laplace equation, which is called Winslow mesh generation system. Accordingly, Equation (2.3) is often called the inhomogeneous TTM system.

Transforming the equation (2.3) into the computational space, one has:

$$\alpha \frac{\partial^2 x}{\partial \xi^2} - 2\beta \frac{\partial^2 x}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 x}{\partial \eta^2} + \delta \left(P \frac{\partial x}{\partial \xi} + Q \frac{\partial x}{\partial \eta} \right) = 0 \quad (2.4a)$$

$$\alpha \frac{\partial^2 y}{\partial \xi^2} - 2\beta \frac{\partial^2 y}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 y}{\partial \eta^2} + \delta \left(P \frac{\partial y}{\partial \xi} + Q \frac{\partial y}{\partial \eta} \right) = 0 \quad (2.4b)$$

where $\alpha = g_{22}$, $\beta = g_{12}$, $\gamma = g_{11}$, $\delta = g$ and g is the metric tensor with

$$g = \begin{vmatrix} (x_\xi^2 + y_\xi^2) & (x_\xi x_\eta + y_\xi y_\eta) \\ (x_\xi x_\eta + y_\xi y_\eta) & (x_\eta^2 + y_\eta^2) \end{vmatrix} \quad (2.4c)$$

The orthogonal condition is: $\beta = g_{12} = g_{21} = 0$.

Using the central difference scheme to discretise the Equation (2.4) and substituting the orthogonal condition into it, we obtain:

$$\begin{aligned} x_{i,j}^n = & \frac{\alpha'}{2\alpha'+2\gamma'}(x_{i-1,j} + x_{i+1,j}) + \frac{\gamma'}{2\alpha'+2\gamma'}(x_{i,j-1} + x_{i,j+1}) + \\ & \frac{0.5\delta'}{2\alpha'+2\gamma'}P(x_{i+1,j} - x_{i-1,j}) + \frac{0.5\delta'}{2\alpha'+2\gamma'}Q(x_{i,j+1} - x_{i,j-1}), \end{aligned} \quad (2.5a)$$

$$\begin{aligned} y_{i,j}^n = & \frac{\alpha'}{2\alpha'+2\gamma'}(y_{i-1,j} + y_{i+1,j}) + \frac{\gamma'}{2\alpha'+2\gamma'}(y_{i,j-1} + y_{i,j+1}) + \\ & \frac{0.5\delta'}{2\alpha'+2\gamma'}P(y_{i+1,j} - y_{i-1,j}) + \frac{0.5\delta'}{2\alpha'+2\gamma'}Q(y_{i,j+1} - y_{i,j-1}), \end{aligned} \quad (2.5b)$$

where superscript “*n*” denotes new value of iteration process, (*i*, *j*) is the index in computation coordinate system and

$$\begin{aligned} \alpha' &= 0.25[(x_{i,j+1} - x_{i,j-1})^2 + (y_{i,j+1} - y_{i,j-1})^2] \\ \gamma' &= 0.25[(x_{i+1,j} - x_{i-1,j})^2 + (y_{i+1,j} - y_{i-1,j})^2] \\ \delta' &= [(x_{i+1,j} - x_{i-1,j})(y_{i,j+1} - y_{i,j-1}) - (x_{i,j+1} - x_{i,j-1})(y_{i+1,j} - y_{i-1,j})]^2 / 16. \end{aligned} \quad (2.5c)$$

2.3.2 RL Mesh Generation System

In the TTM system, the following control functions are defined for the orthogonal mesh generation:

$$P = \frac{1}{h_\xi h_\eta} f_\xi, \quad Q = \frac{1}{h_\xi h_\eta} \left(\frac{1}{f}\right)_\eta \quad (2.6)$$

where h_ξ and h_η are scale factors and $h_\xi = g_{11}^{1/2}$, $h_\eta = g_{22}^{1/2}$; and f is the aspect ratio (also called the distortion function) and is defined as: $f = \frac{h_\eta}{h_\xi} = \left(\frac{x_\eta^2 + y_\eta^2}{x_\xi^2 + y_\xi^2}\right)^{1/2}$.

The substitution of Equation (2.6) into (2.5) and the rearrangement leads to a covariant Laplace equation system proposed by Ryskin and Leal (1983) for the orthogonal mesh generation.

$$\frac{\partial}{\partial \xi} \left(f \frac{\partial x}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(\frac{1}{f} \frac{\partial x}{\partial \eta} \right) = 0 \quad (2.7a)$$

$$\frac{\partial}{\partial \xi} \left(f \frac{\partial y}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(\frac{1}{f} \frac{\partial y}{\partial \eta} \right) = 0 \quad (2.7b)$$

Using the central difference scheme to discretize Equation (2.7) at one typical grid point (i, j) , one can obtain:

$$F_{i,j} x_{i,j} = f_{i+1/2,j} x_{i+1,j} + f_{i-1/2,j} x_{i-1,j} + \frac{1}{f_{i,j+1/2}} x_{i,j+1} + \frac{1}{f_{i,j-1/2}} x_{i,j-1} \quad (2.8a)$$

$$F_{i,j} y_{i,j} = f_{i+1/2,j} y_{i+1,j} + f_{i-1/2,j} y_{i-1,j} + \frac{1}{f_{i,j+1/2}} y_{i,j+1} + \frac{1}{f_{i,j-1/2}} y_{i,j-1} \quad (2.8b)$$

$$\text{where } F_{i,j} = f_{i+1/2,j} + f_{i-1/2,j} + \frac{1}{f_{i,j+1/2}} + \frac{1}{f_{i,j-1/2}}.$$

2.3.3 Improved TTM and RL Systems with Smoothness Function

In order to achieve a good balance of the orthogonality and smoothness without the mesh distortion and overlapping in geometrically complex domains, Zhang et al. (2004) proposed two methods. The second method will be introduced in the next section.

In the first method, the original TTM system and the original RL system are modified and improved by introducing an *effect-control* factor.

The improved TTM system is as follows:

$$\alpha \frac{\partial^2 x}{\partial \xi^2} - 2\beta \frac{\partial^2 x}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 x}{\partial \eta^2} + \delta [(1 - r_p)P \frac{\partial x}{\partial \xi} + (1 - r_q)Q \frac{\partial x}{\partial \eta}] = 0 \quad (2.9a)$$

$$\alpha \frac{\partial^2 y}{\partial \xi^2} - 2\beta \frac{\partial^2 y}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 y}{\partial \eta^2} + \delta [(1 - r_p)P \frac{\partial y}{\partial \xi} + (1 - r_q)Q \frac{\partial y}{\partial \eta}] = 0 \quad (2.9b)$$

The improved RL system becomes

$$\frac{\partial}{\partial \xi} \left(f \frac{\partial x}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(\frac{1}{f} \frac{\partial x}{\partial \eta} \right) + P_x + Q_x = 0 \quad (2.10a)$$

$$\frac{\partial}{\partial \xi} \left(f \frac{\partial y}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(\frac{1}{f} \frac{\partial y}{\partial \eta} \right) + P_y + Q_y = 0 \quad (2.10b)$$

where P_x, Q_x, P_y and Q_y are called smoothness control functions and defined by:

$$(P_x, P_y) = -r_p f_\xi \frac{\partial(x, y)}{\partial \xi} \quad (2.11a)$$

$$(Q_x, Q_y) = -r_q \left(\frac{1}{f} \right)_\eta \frac{\partial(x, y)}{\partial \eta} \quad (2.11b)$$

where r_p ($\in [0,1]$) and r_q ($\in [0,1]$) are *effect-control* factors in the ξ and η directions, respectively.

The following formulas are constructed to evaluate the effect control factors.

$$r_p = \frac{|\bar{h}_\xi - h_\xi|}{\bar{h}_\xi} \quad (2.12a)$$

$$r_q = \frac{|\bar{h}_\eta - h_\eta|}{\bar{h}_\eta} \quad (2.12b)$$

where \bar{h}_ξ and \bar{h}_η are the locally averaged scale factors in the ξ and η directions respectively.

The *effect-control* factor is constructed to adjust the balance between the smoothness and the orthogonality automatically for the whole domain. The local smoothness increases, and the local orthogonality decreases with the increase of the effect-control factor, and vice versa. With this mechanism, the proposed method takes the advantages from both conformal mapping and orthogonal mapping while avoiding their defects.

2.3.4 Improved RL Systems with Contribution Factors

For the second method, the original RL system are modified and improved by introducing a contribution factor.

The improved RL system has the following form:

$$F_{i,j} x_{i,j} = f_{i+1/2,j} c_{i+1,j} x_{i+1,j} + f_{i-1/2,j} c_{i-1,j} x_{i-1,j} + \frac{c_{i,j+1}}{f_{i,j+1/2}} x_{i,j+1} + \frac{c_{i,j-1}}{f_{i,j-1/2}} x_{i,j-1} \quad (2.13a)$$

$$F_{i,j}y_{i,j} = f_{i+1/2,j}c_{i+1,j}y_{i+1,j} + f_{i-1/2,j}c_{i-1,j}y_{i-1,j} + \frac{c_{i,j+1}}{f_{i,j+1/2}}y_{i,j+1} + \frac{c_{i,j-1}}{f_{i,j-1/2}}y_{i,j-1} \quad (2.13b)$$

where $F_{i,j} = f_{i+1/2,j}c_{i+1,j} + f_{i-1/2,j}c_{i-1,j} + \frac{c_{i,j+1}}{f_{i,j+1/2}} + \frac{c_{i,j-1}}{f_{i,j-1/2}}$ and $c_{i,j}$ is the contribution factor of grid point (i, j) .

In Equations (2.13), the contribution factor is defined as follows:

$$c_{i+1,j} = (d_{i+1,j})^\alpha = (\sqrt{(x_{i,j} - x_{i+1,j})^2 + (y_{i,j} - y_{i+1,j})^2})^\alpha \quad (2.14a)$$

$$c_{i-1,j} = (d_{i-1,j})^\alpha = (\sqrt{(x_{i,j} - x_{i-1,j})^2 + (y_{i,j} - y_{i-1,j})^2})^\alpha \quad (2.14b)$$

$$c_{i,j+1} = (d_{i,j+1})^\alpha = (\sqrt{(x_{i,j} - x_{i,j+1})^2 + (y_{i,j} - y_{i,j+1})^2})^\alpha \quad (2.14c)$$

$$c_{i,j-1} = (d_{i,j-1})^\alpha = (\sqrt{(x_{i,j} - x_{i,j-1})^2 + (y_{i,j} - y_{i,j-1})^2})^\alpha \quad (2.14d)$$

where $d_{i+1,j}$ is the distance between point (i, j) and point $(i+1, j)$ and $\alpha \in [0, 1]$ is a parameter.

The contribution factor is evaluated by the distance of the grid point (i, j) and its neighbors. The neighbor with longer distance has larger contribution factor, while the neighbor with smaller distance has smaller contribution factor. For one grid point (i, j) , if the distortion function f of one neighbor is high, which implies that the distance between them is small, then the contribution factor of that neighbor is also small, which will decrease the contribution of that node. This mechanism can confine the vicious cycle and thus make the mesh smoother.

With the center difference scheme, the following relationships can be obtained:

$$c_{i+1,j} = (d_{i+1,j})^\alpha = [(h_\xi)_{i+1/2,j}]^\alpha \quad (2.15a)$$

$$c_{i-1,j} = (d_{i-1,j})^\alpha = [(h_\xi)_{i-1/2,j}]^\alpha \quad (2.15b)$$

$$c_{i,j+1} = (d_{i,j+1})^\alpha = [(h_\eta)_{i,j+1/2}]^\alpha \quad (2.15c)$$

$$c_{i,j-1} = (d_{i,j-1})^\alpha = [(h_\eta)_{i,j-1/2}]^\alpha \quad (2.15d)$$

Then this improved RL system can be rewritten as the following form:

$$\frac{\partial}{\partial \xi} (h_\xi^\alpha \cdot f \frac{\partial x}{\partial \xi}) + \frac{\partial}{\partial \eta} (h_\eta^\alpha \cdot \frac{1}{f} \frac{\partial x}{\partial \eta}) = 0 \quad (2.16a)$$

$$\frac{\partial}{\partial \xi} (h_\xi^\alpha \cdot f \frac{\partial y}{\partial \xi}) + \frac{\partial}{\partial \eta} (h_\eta^\alpha \cdot f \frac{1}{f} \frac{\partial y}{\partial \eta}) = 0 \quad (2.16b)$$

Zhang et al. (2006a) analyzed the above equation (2.16) and proposed an automatic smoothness control

Zhang et al. (2006a) proposed an automatic smoothness control mechanism based on five types of smoothness conditions and includes the self-adjustment mechanism and the auto-evaluation mechanism for the empirical parameter α .

$$(DLS)_\xi^{Local} |_{i,j} = \frac{|(h_\xi)_{i,j} - (\bar{h}_\xi)_{i,j}|}{(\bar{h}_\xi)_{i,j}} \quad (2.17a)$$

$$(DLS)_\eta^{Local} |_{i,j} = \frac{|(h_\eta)_{i,j} - (\bar{h}_\eta)_{i,j}|}{(\bar{h}_\eta)_{i,j}} \quad (2.17b)$$

$$(DLS)_\xi^{Global} |_{i,j} = \frac{|(h_\xi)_{i,j} - (\bar{h}_\xi)_i|}{(\bar{h}_\xi)_i} \quad (2.17c)$$

$$(DLS)_\eta^{Global} |_{i,j} = \frac{|(h_\eta)_{i,j} - (\bar{h}_\eta)_j|}{(\bar{h}_\eta)_j} \quad (2.17d)$$

$$(DLS)_{i,j} = \frac{|(h_\xi)_{i,j} - (h_\eta)_{i,j}|}{\max[(h_\xi)_{i,j}, (h_\eta)_{i,j}]} \quad (2.17e)$$

where DLS represents the Deviation from the Local Smoothness Conditions; $(\bar{h}_\xi)_{i,j}$ and $(\bar{h}_\eta)_{i,j}$ are the local averaged scale factors, $(\bar{h}_\xi)_i$ and $(\bar{h}_\eta)_j$ are the global averaged scale factors; the superscript “*Local*” denotes the use of the local averaged scale factors, and the superscript “*Global*” denotes the use of the global averaged scale factors.

The Averaged Deviations from local Smoothness conditions (ADS) is defined as the algebraic average of the above five indicators.

$$(ADS)_{i,j} = \left[\frac{(DLS)_\xi^{Local} + (DLS)_\eta^{Local} + (DLS)_\xi^{Global} + (DLS)_\eta^{Global} + (DLS)_{i,j}}{5} \right]_{i,j} \quad (2.18)$$

Then the parameter α is evaluated by

$$\alpha = \max[\min(ADS_{i,j}, 1)] \quad (2.19)$$

In CCHE-MESH, this automatic smoothness control method is referred as "RL Orthogonal Mesh with auto smoothness controls [1]".

2.3.5 Improved RL Systems with Controls on Distortion Function

Zhang et al. (2006b) proposed a method to control the distortion function of the RL system to improve mesh quality in geometrically complex domains. In this method, the distortion function is determined by both the scale factors and the averaged scale factors of the constant mesh lines. Two adjustable parameters are used to control the local balance of the orthogonality and the smoothness.

The distortion function is evaluated by

$$\overline{f}_{i,j} = \frac{(\bar{h}_\eta)_j \cdot r_\eta + (h_\eta)_{i,j} \cdot (1 - r_\eta)}{(\bar{h}_\xi)_i \cdot r_\xi + (h_\xi)_{i,j} \cdot (1 - r_\xi)} \quad (2.20)$$

where r_η and r_ξ are two adjustable parameters within the range of [0, 1] to control the ratio between the averaged scale factors and the local scale factors.

The parameters r_η and r_ξ can be evaluated by the ratio of the difference and the sum of the local scale factors and the averaged scale factor in the corresponding directions.

$$[r_\xi]_{i,j} = \frac{|(h_\xi)_{i,j} - (\bar{h}_\xi)_i|}{(h_\xi)_{i,j} + (\bar{h}_\xi)_i} \quad (2.21a)$$

$$[r_\eta]_{i,j} = \frac{|(h_\eta)_{i,j} - (\bar{h}_\eta)_j|}{(h_\eta)_{i,j} + (\bar{h}_\eta)_j} \quad (2.21b)$$

In CCHE-MESH, this method is referred as "RL Orthogonal Mesh with auto smoothness controls [2]".

2.3.6 Improved RL Adaptive Mesh Generation System

Zhang et al. (2007) derived a two-dimensional adaptive elliptic mesh generation system from the Ryskin and Leal (RL) orthogonal mesh generation system based on the orthogonal condition (orthogonality) and the cell area equal-distribution principle (adaptivity). he proposed generation system takes into account not only the mesh orthogonality and adaptivity but also the mesh smoothness by adopting a method that the distortion functions is

determined by both the scale factors and the averaged scale factors of the constant mesh lines. (zhang et al., 2006b).

The conservative form of this adaptive system is as follows:

$$\frac{\partial}{\partial \xi} (f \cdot J_w \cdot \frac{\partial x}{\partial \xi}) + \frac{\partial}{\partial \eta} (\frac{1}{f} \cdot J_w \cdot \frac{\partial x}{\partial \eta}) = 0 \quad (2.22a)$$

$$\frac{\partial}{\partial \xi} (f \cdot J_w \cdot \frac{\partial y}{\partial \xi}) + \frac{\partial}{\partial \eta} (\frac{1}{f} \cdot J_w \cdot \frac{\partial y}{\partial \eta}) = 0 \quad (2.22b)$$

where J denotes the Jacobian and w is the weighting function.

Alternatively, we have the non-conservative form:

$$\frac{\partial}{\partial \xi} (f \cdot \frac{\partial x}{\partial \xi}) + \frac{\partial}{\partial \eta} (\frac{1}{f} \cdot \frac{\partial x}{\partial \eta}) + P = 0 \quad (2.23a)$$

$$\frac{\partial}{\partial \xi} (f \cdot \frac{\partial y}{\partial \xi}) + \frac{\partial}{\partial \eta} (\frac{1}{f} \cdot \frac{\partial y}{\partial \eta}) + Q = 0 \quad (2.23b)$$

Where P and Q are called control functions.

$$P = (\frac{J_\xi}{J} + \frac{w_\xi}{w}) f \cdot x_\xi + (\frac{J_\eta}{J} + \frac{w_\eta}{w}) \frac{1}{f} \cdot x_\eta \quad (2.24a)$$

$$Q = (\frac{J_\xi}{J} + \frac{w_\xi}{w}) f \cdot y_\xi + (\frac{J_\eta}{J} + \frac{w_\eta}{w}) \frac{1}{f} \cdot y_\eta \quad (2.24b)$$

2.4 Mesh Evaluation

The quality of mesh is evaluated quantitatively by several indicators, such as Maximum Deviation Orthogonality (MDO), Averaged Deviation from Orthogonality (ADO), Maximum grid Aspect Ratio (MAR), and Averaged grid Aspect Ratio (AAR).

The MDO and ADO, which are used to evaluate the orthogonality of a mesh, are defined as

$$MDO = \max(\theta_{i,j}) \quad (2.25a)$$

$$ADO = \frac{1}{(ni-2)(nj-2)} \sum_2^{ni-1} \sum_2^{nj-1} \max(\theta_{i,j}) \quad (2.25b)$$

where ni and nj are the maximum number of mesh lines in ξ and η directions respectively; and θ is defined as

$$\theta_{i,j} = |\arccos\left(\frac{g_{12}}{h_\xi h_\eta}\right)_{i,j} - 90| \quad (2.26)$$

The MAR and AAR, which are used to evaluate the smoothness of a mesh, are defined as

$$MAR = \max[\max(f_{i,j}, \frac{1}{f_{i,j}})] \quad (2.27a)$$

$$AAR = \frac{1}{(ni-2)} \frac{1}{(nj-2)} \sum_2^{ni-1} \sum_2^{nj-1} \max[\max(f_{i,j}, \frac{1}{f_{i,j}})] \quad (2.27b)$$

3 BED INTERPOLATION

3.1 Introduction

To represent a physical domain, not only the planar two-dimensional discretization but also the bed interpolation is needed for a computational mesh. Bed interpolation is a process of interpolating bed elevation from the topology or bathymetrical database onto the mesh nodes. Usually the resolution and the distribution of the interpolation points in the database are different from those of the mesh nodes. The resolution and distribution differences between the database and the mesh would make an accurate interpolation difficult even impossible.

Obviously the database plays a very important role in the bed interpolation. In CCHE-MESH, the database is grouped into two categories: the random database and the structured database. The structured database is well organized with the cross-sectional measured data, while in the random database the data points are randomly placed. In CCHE-MESH, different interpolation algorithms were developed for different types of database.

3.2 Interpolation Methods

In CCHE-MESH, there are two interpolation methods. The first one is the weighted-average method.

3.2.1 Inverse Distance Weighting Method

Given a set of interpolation points $Z_i (i = 1, 2, \dots, n)$, the bed elevation at the grid point G can be obtained using the following formula:

$$Z_G = \frac{\sum_{i=1}^n w_i \cdot Z_i}{\sum w_i} \quad (3.1)$$

Where w_i is the weighting factor of the interpolation point i .

One famous weighted-average method is the inverse distance weighting method (IDW), in which the weighting factor is proportional to the inverse power of the distance between the interpolation point and the gird point. That is,

$$Z_G = \frac{\sum_{i=1}^n \frac{Z_i}{d_i^p}}{\sum \frac{1}{d_i^p}} \quad (3.2)$$

In Equation (3.2), d_i is the distance between the interpolation point i and the gird point. the exponential parameter p can adjust the weighting further. In CCHE-MESH, $p = 1$.

3.2.2 Planar Interpolation Method

The second interpolation method is the planar interpolation based on a triangle plane. As shown in Figure 3-1, the bed elevation at grid point G can be obtained.

$$a = y_1 \cdot (z_2 - z_3) + y_2 \cdot (z_3 - z_1) + y_3 \cdot (z_1 - z_2) \quad (3.3a)$$

$$b = z_1 \cdot (x_2 - x_3) + z_2 \cdot (x_3 - x_1) + z_3 \cdot (x_1 - x_2) \quad (3.3b)$$

$$c = x_1 \cdot (y_2 - y_3) + x_2 \cdot (y_3 - y_1) + x_3 \cdot (y_1 - y_2) \quad (3.3c)$$

$$d = -x_1 \cdot (y_2 z_3 - y_3 z_2) - x_2 \cdot (y_3 z_1 - y_1 z_3) - x_3 \cdot (y_1 z_2 - y_2 z_1) \quad (3.3d)$$

$$Z_G = -(a \cdot x_G + b \cdot y_G + d) / c \quad (\text{if } c \neq 0) \quad (3.3e)$$

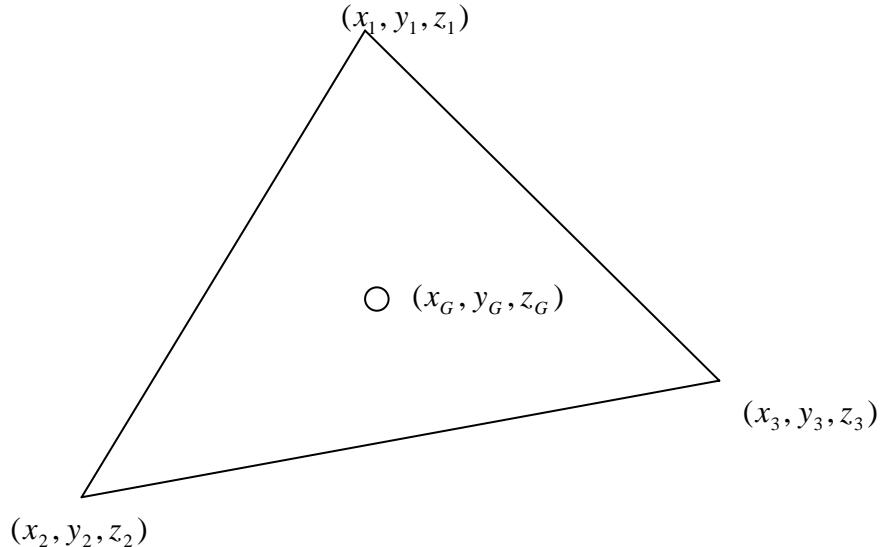


Figure 3-1 A triangle plane

3.3 Interpolation Algorithms

In CCHE-MESH Mesh Generator, there are two algorithms for the random database and the structured database, respectively.

3.3.1 Random Database

For the random database, triangulation algorithm is used. The database is first triangulated using the Delaunay triangulation method. Then the triangle contains the interpolation mesh node is identified. In this interpolation triangle, the planar interpolation method is used to get the bed elevation. Figure 3-2 shows an example of the triangulation of the database.

The triangulation interpolation is accurate and especially suitable for the coarse database. The drawback lies in the huge computational efforts it needs. The computational efforts increase non-linearly (>1) with the increase of the number of the data points.

In CCHE-MESH, two algorithms are used to interpolate the bed elevation from the random database. The first one is the global triangulation and the second is the local triangulation. For the first method, the whole database is triangulated using the incremental triangulation algorithm; while for the second one, only a certain region which contains the interpoataed

mesh node will be triangulated. Obviously, the local triangulation method is faster, but the difficulty lies in the determination of the local triangulation region.

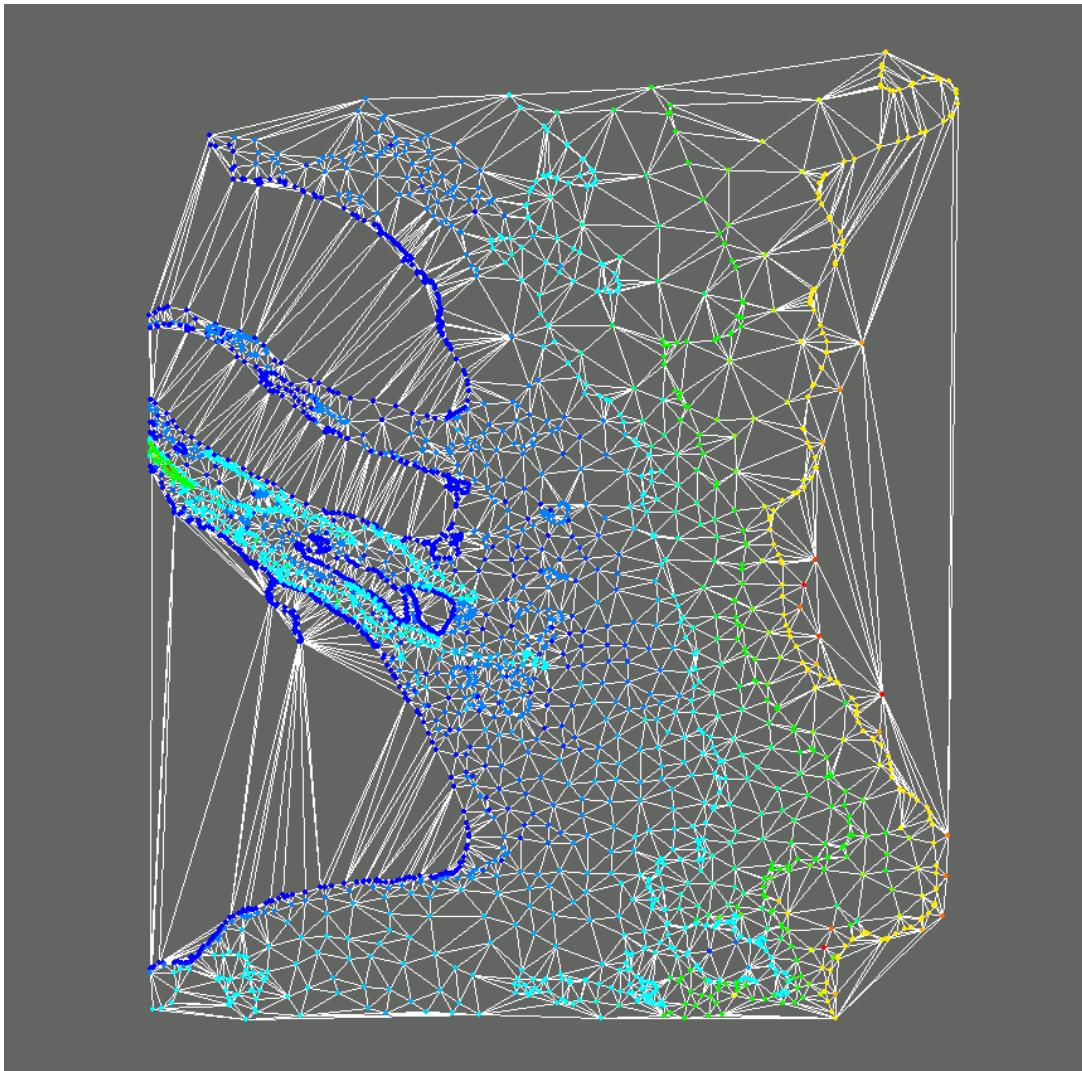


Figure 3-3 Triangulation of database

3.3.2 Structured Database

For the structured database, to make full use of its structured characteristic, a special algorithm was developed. In CCHE-MESH, this algorithm is called structured interpolation.

In the structured interpolation, the cross-sectional survey database is refined to improve the accuracy of the interpolation by considering the flow direction. The refinement algorithm is as follows:

- Refine the data points in the transverse direction along each cross section using linear interpolation;
- Divide each cross section into three parts, left bank, main channel, and right bank.
- Redistribute the equal number of points in these three parts;
- Refine the database in the longitudinal direction between cross sections. Between cross sections, the linear interpolation will be conducted between the corresponding parts of each cross section.

After refinement, the database contains refined cross sections with equal number of the data points. Then a structured mesh is formed based on these cross sections. After the quadrilateral that contains the interpolation mesh node is identified, the IDW method will be used to interpolate the bed elevation.

The structured interpolation is also more stable than the random interpolation, but it requires a structured database (measured cross sectional data).

4 GRAPHICAL USERS INTERFACE

4.1 Users Interface

In this section, the Graphical Users Interface of the CCHE-MESH will be introduced. You will get familiar with each functional component.

4.1.1 Main Window

Figure 4-1 shows the main window of the CCHE-MESH. It is composed of a **Title Bar** that displays the path of the file, a **Menu Bar**, **Toolbars**, **Status Bar** that displays the tips and the probe information, a **Project view panel**, and the **Plot Area**.

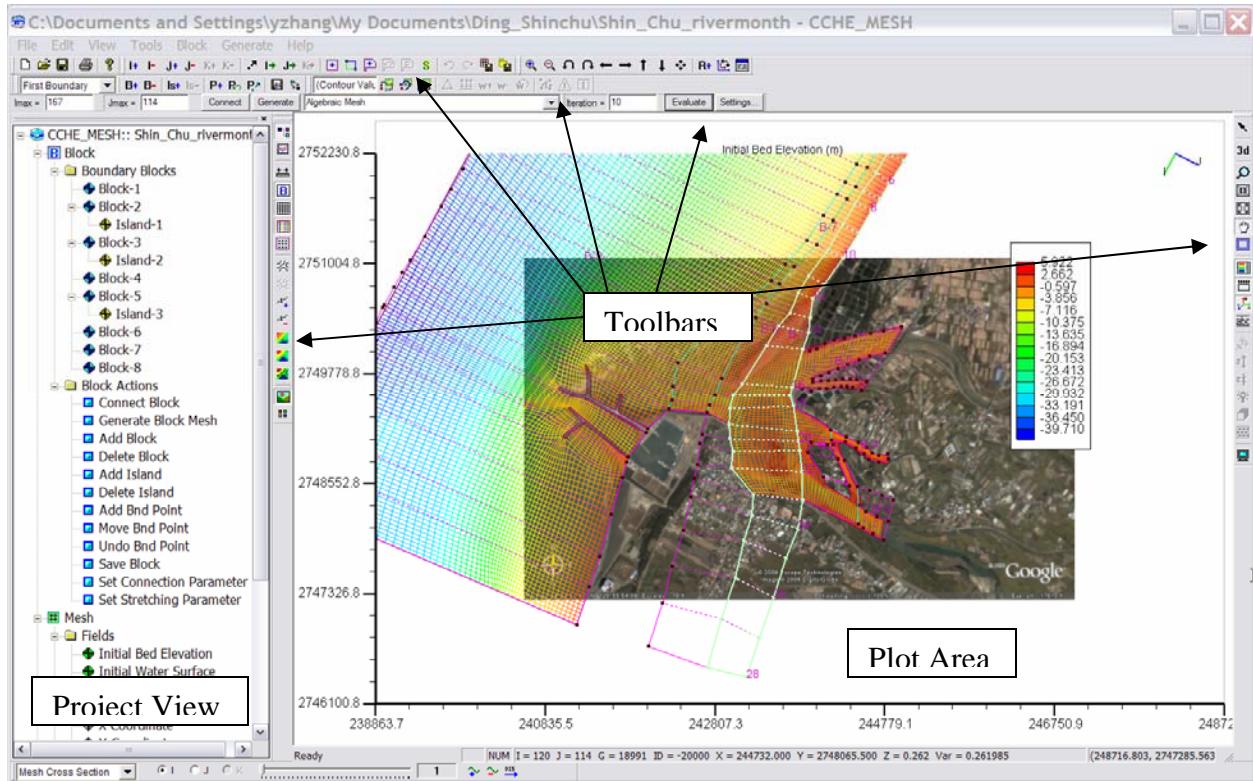


Figure 4-1 Main Window

4.1.2 Toolbars

There are seven toolbars, namely, **Standard** toolbar, **Mesh Generation** toolbar, **Mesh Editing** toolbar, **Block Editing** toolbar, **Tools** toolbar, **View** toolbar and **View Tool** toolbar.

4.1.2.1 Standard Toolbar



New : Use this button to close the current workspace and create a new workspace.

Open : Use this button to open a mesh file (*.geo").

Save : Use this button to save the current workspace.

Print : Use this button to print the current plot.

Help ?: Use this button to get version information and contact information for help.

4.1.2.2 Mesh Generation Toolbar



I_{max}: Input the maximum number of I lines.

J_{max}: Input the maximum number of J Lines.

Connect: Use this button to connect the blocks with mesh.

Generate: Use this button to generate meshes.

Iteration: Input the iteration number.

Evaluate: Use this button to evaluate the mesh.

Settings: Use this button to activate the parameter dialog and set the generation parameters.

4.1.2.3 Mesh Editing Toolbar



Add I Line : Use this button to add a mesh line in I direction.

Delete I Line : Use this button to delete a mesh line in I direction.

Add J Line : Use this button to add a mesh line in J direction.

Delete J Line : Use this button to delete a mesh line in J direction.

Add K Line : Use this button to add a mesh line in K direction.

Delete K Line : Use this button to delete a mesh line in K direction.

Move Mesh Node : Use this button to move a mesh node.

Move I Line : Use this button to move a mesh line in I direction.

Move J Line : Use this button to move a mesh line in J direction.

Move K Line : Use this button to move a mesh line in K direction.

Extend mesh from starting J line : Use this button to extend mesh from the starting J line.

Extend mesh from ending J line : Use this button to extend mesh from the ending J line.

Define Rectangular Region : Use this button to define a rectangular region.

Select Whole Domain : Use this button to select the whole domain

Define a Polygon : Use this button to define a polygon region and set a value for this region or get bed change information for this region.

Undo Polygon Point : Use this button to undo a polygon point when defining a polygon.

Show Polygon : Use this button to show or hide polygons.

Smooth Bed : Use this button to smooth the bed.

Undo : Use this button to undo the previous actions.

Restore : Use this button to restore the previous actions.

Save Changes : Save the changes into the current data set.

Save Changes as : Save the changes into a new data set.

4.1.2.4 Block Editing Toolbar



First Boundary : Select the boundary you want to define.

Add Block : Use this button to add a block.

Delete Block : Use this button to delete a block.

Add Island : Use this button to add a island.

Delete Island : Use this button to delete a island.

Add Bnd Point : Use this button to add a bnd point.

Undo Bnd Point : Use this button to delete the last bnd point.

Move Bnd Point : Use this button to move a bnd point.

Save Block : Use this button to save the blocks into a file.

4.1.2.5 Tools Toolbar



(Contour Valu: Input contour value when extracting contour data from an image.

Extract Data From Image: Use this button to start to extract data from image.

Undo Data Point: Use this button to undo the previous extracted point.

Save Extracted Data: Use this button to save the extracted data into a file.

Triangulate Database: Use this button to triangulate the topography database.

Refine Database: Use this button to refine the topography database.

Define Wet Line: Use this button to define a wet line for refinement.

Delete Wet Line: Use this button to delete a wet line.

Undo Point on Wet Line: Use this button to undo a point on the wet line.

Random Interpolation: Use this button to do the random interpolation.

Triangulation Interpolation: Use this button to do the triangulation interpolation.

Structured Interpolation: Use this button to do the structured interpolation.

4.1.2.6 Image Editing Toolbar



Zoom In: Use this button to zoom in the image.

Zoom Out: Use this button to zoom out the image.

Rotate Anticlockwise: Use this button to rotate the image anti-clockwise.

Rotate Clockwise: Use this button to rotate the image clockwise.

Move Left: Use this button to move the image toward left.

Move Right: Use this button to move the image toward right.

- Move Up** : Use this button to move the image upward.
- Move Down** : Use this button to move the image downward.
- Pan** : Use this button to pan the image.
- Coordinate Transformation** : Use this button to define two points to transform the coordinate system of the image.
- Save Transformation** : Use this button to save the transformation.
- Settings** : Use this button to set the parameters of the transformation.

4.1.2.7 View Toolbar



Project view : Use this button to show or hide Project view.

XY Plot View : Use this button to show or hide XY plot view.

Block : Use this button to show or hide blocks.

Mesh : Use this button to show or hide mesh.

Colored Mesh : Use this button to show or hide colored mesh.

Contour Line : Use this button to show or hide contour line.

Colored Contour Line : Use this button to show or hide colored contour line.

Flood Shading : Use this button to show or hide flood shading.

Contour Shading : Use this button to show or hide contour shading.

Contour Shading + Contour Line : Use this button to show or hide contour with flood + lines.

Image : Use this button to show or hide the image.

Scatter Points : Use this button to show or hide the scatter points.

4.1.2.8 View Tool Toolbar



Select : Use this button to select Title, Text, Time and Legend and move them.

3d : Use this button to show/Hide 3D view.

Zoom : Use this button to zoom in.

Incremental Zoom In : Use this button to zoom in incrementally.

Incremental Zoom Out : Use this button to zoom out incrementally.

Pan : Use this button to pan.

Full Size : Use this button to restore to full size view.

Legend : Use this button to show or hide legend.

Title : Use this button to show or hide title.

Axis : Use this button to show or hide axis.

Text : Use this button to add and edit the texts.

Rotate : Use this button to rotate the view.

Increase Z : Use this button to increase the scale in Z direction.

Decrease Z : Use this button to decrease the scale in Z direction.

Light : Use this button to enable or disable light effects.

Frame : Use this button to show or hide 3D frame.

Texture : Use this button to enable or disable texture effects.

Options : Use this button to set view options.

4.1.3 Menus

Each menu item is quite self-explained. Most of menus items have corresponding toolbar buttons for easy access.

4.1.3.1 File Menu

This menu deals with the files input and output (I/O).

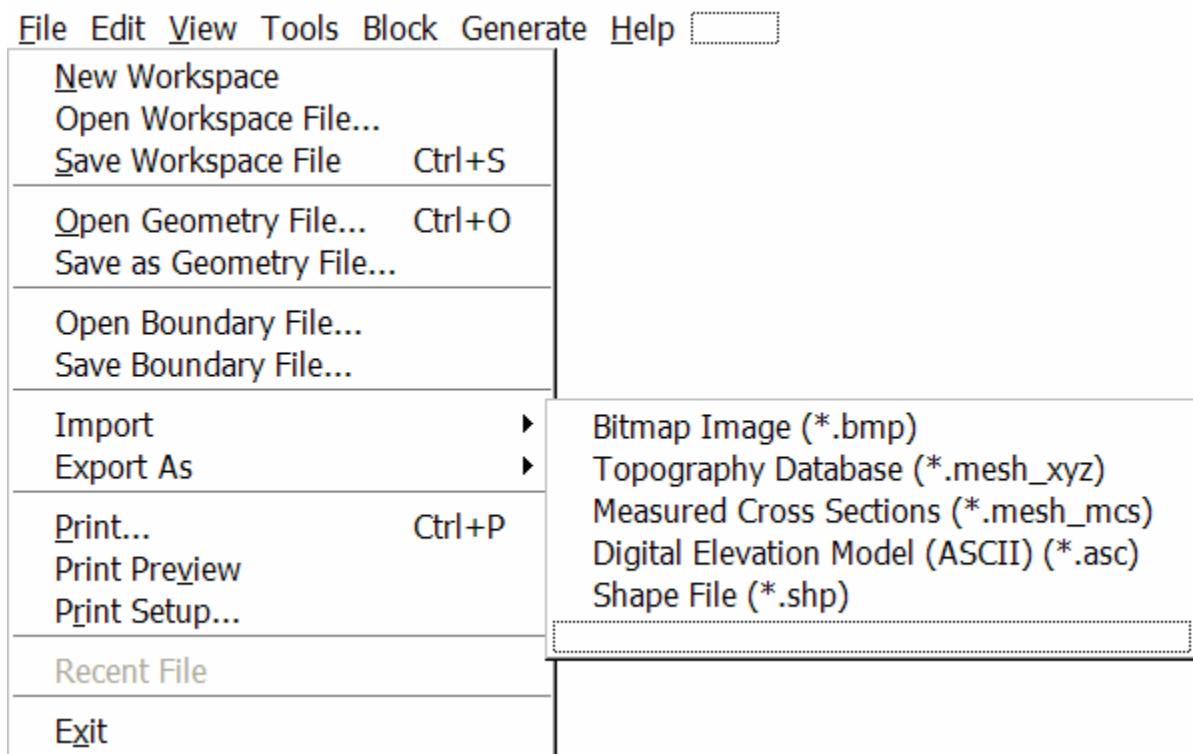


Figure 4-2

4.1.3.2 Edit Menu

This menu is for mesh editing.

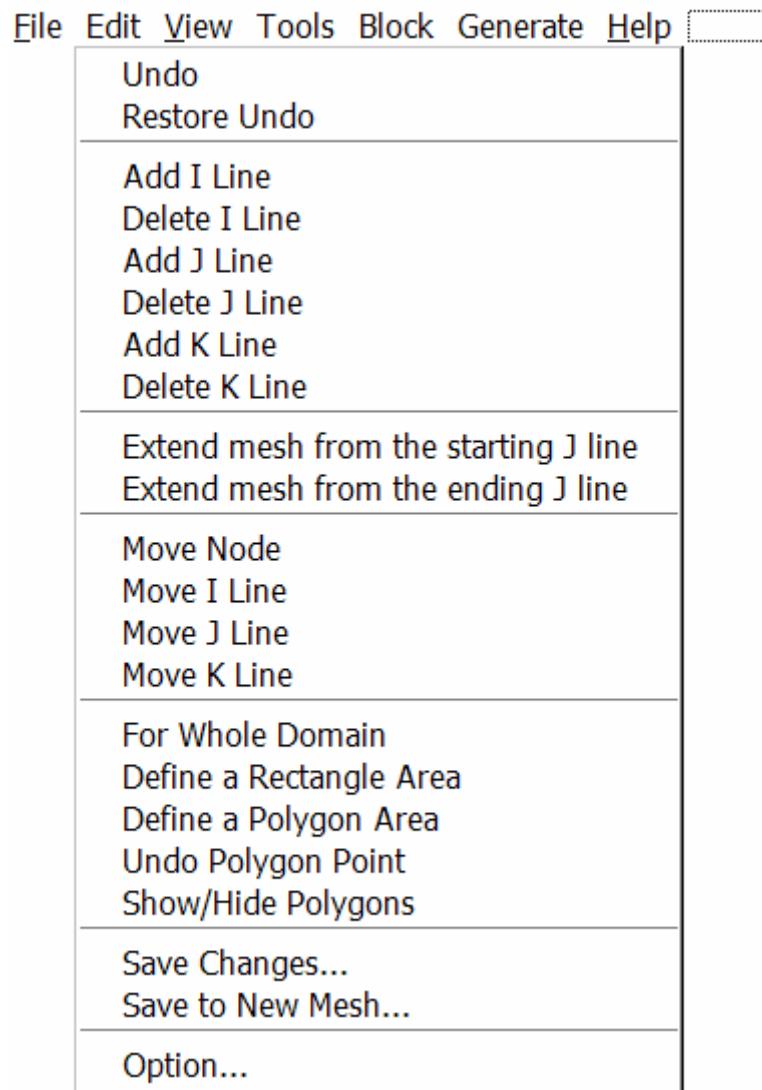


Figure 4-3

4.1.3.3 View Menu

Use this menu to customize your views.

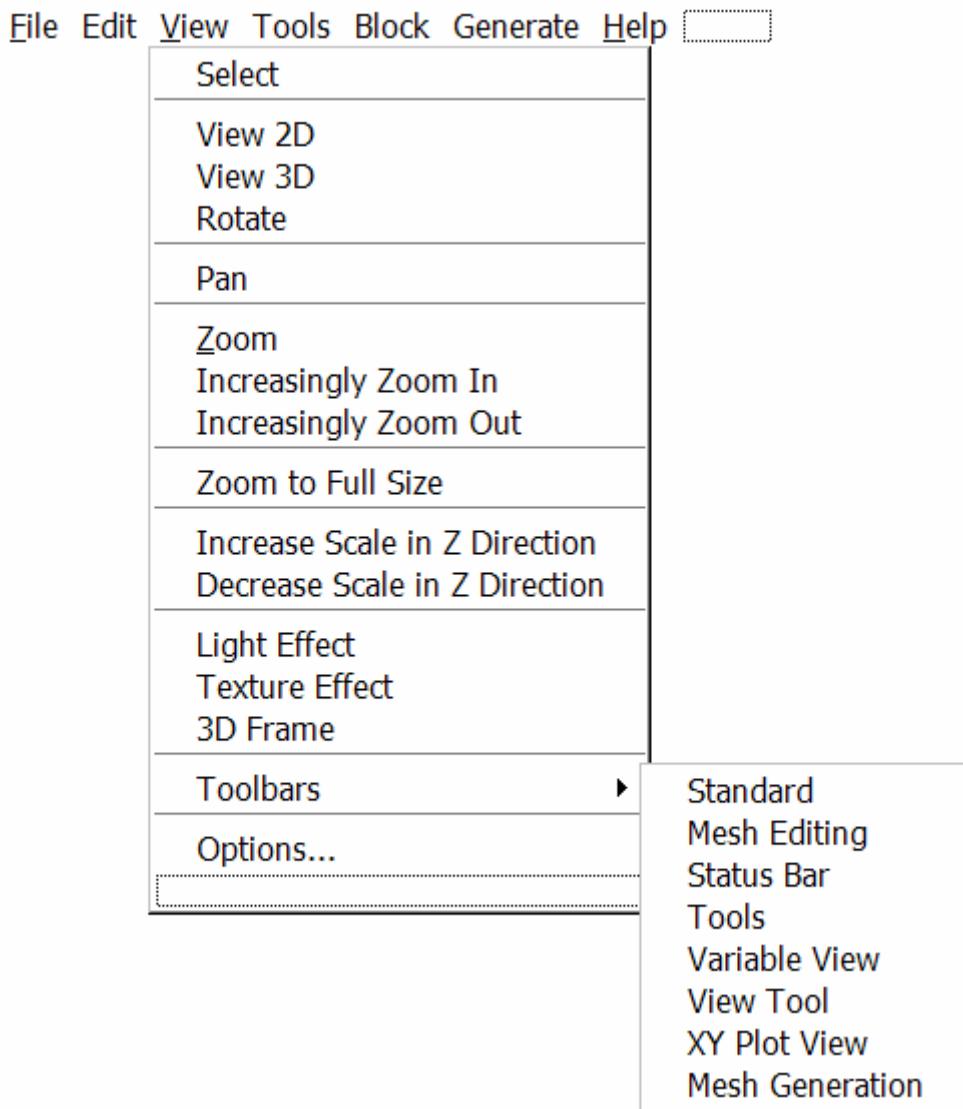


Figure 4-4

4.1.3.4 Tools Menu

This menu mainly contains four tools: tools for topography database, tools for topography image, tools for bed interpolation, and tools for defining effect area.

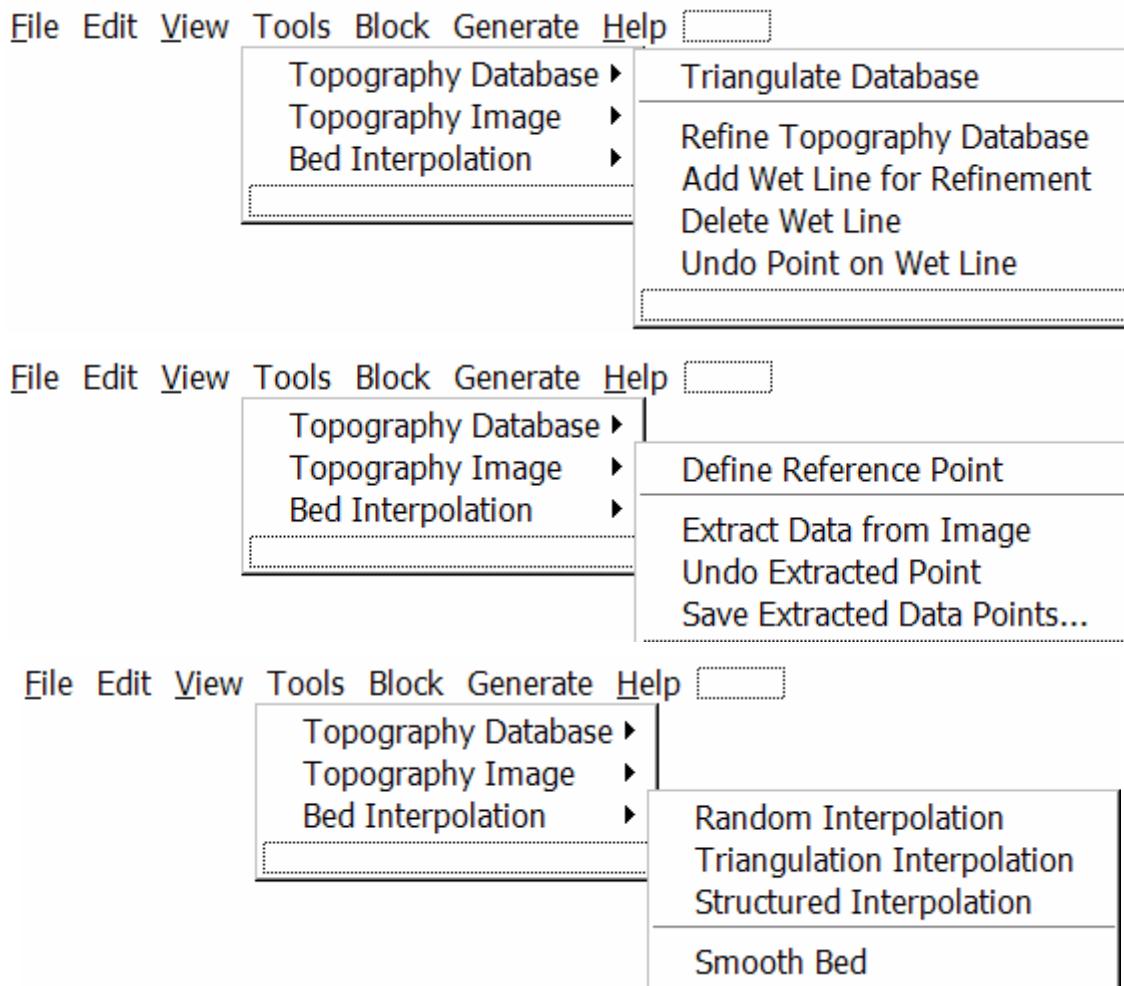


Figure 4-5

4.1.3.5 Block Menu

Use this menu to create blocks.

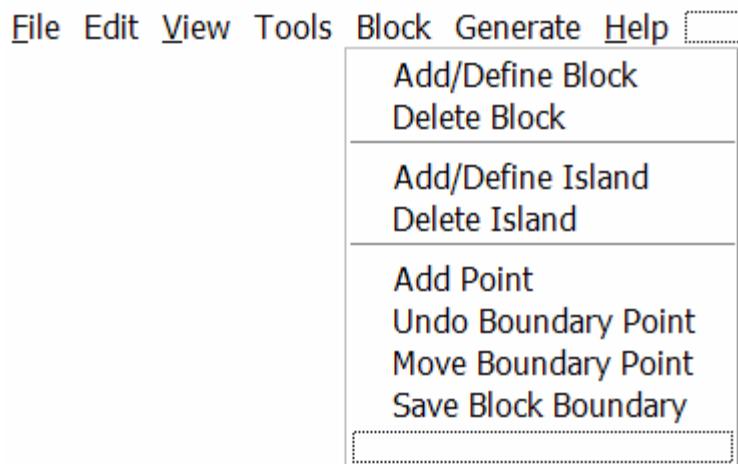


Figure 4-6

4.1.3.6 Generate Menu

Use this menu to generate meshes.

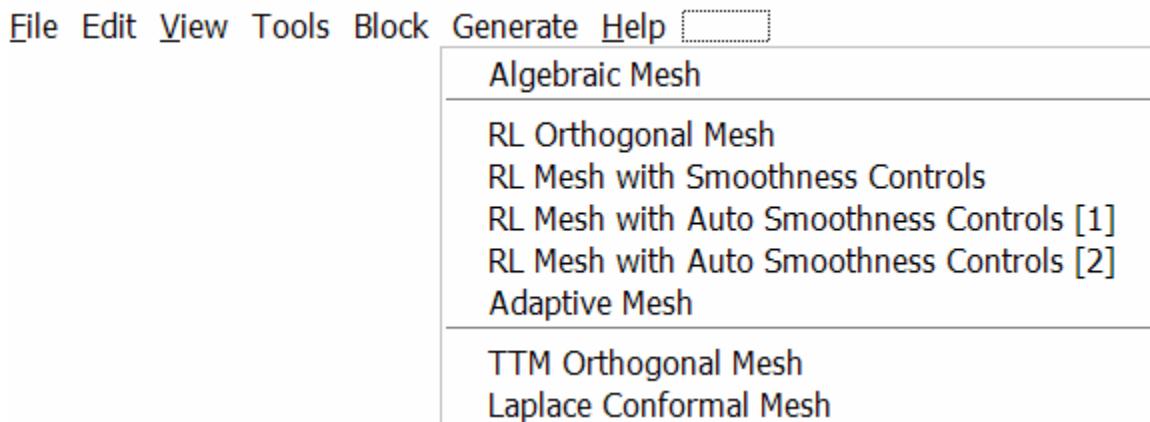
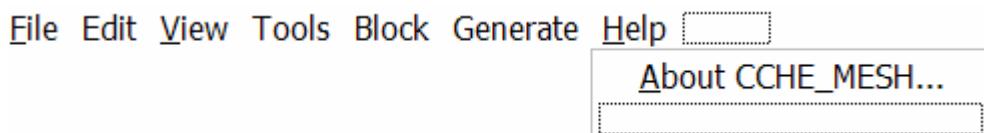


Figure 4-7

4.1.3.7 Help



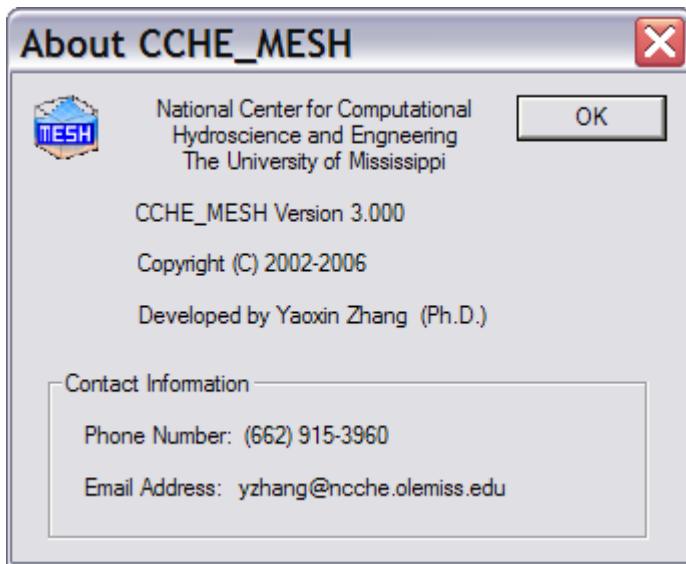


Figure 4-8

4.2 File Management

The CCHE-MESH integrated a simple file management system. For this system, the users are always working with a mesh workspace (*.mesh_wsp). A mesh workspace contains all types of supported files and the view settings.

In this version, the CCHE-MESH supports the following file types: the CCHE geometry file (*.geo), the block boundary file (*.mesh_mb), the 24-bit Bitmap image file, the topography database file (*.mesh_xyz), the measured cross section file (*.mesh_mcs), the ASCII coded Digital Elevation Model file (*.asc), the shape files (*.shp).

The users can start to use the CCHE-MESH with three options as follows:

1. **Create a new workspace** by selecting **New Workspace** from **File** menu or from **Standard** toolbar, and then load the supported files into the workspace. The CCHE-MESH will copy all the files you loaded into the directory where your workspace was created. In case you are working with an existing workspace, now you want to create a new one, the CCHE-MESH will ask for your confirmation. The name of the workspace will be displayed on the **Title** bar.

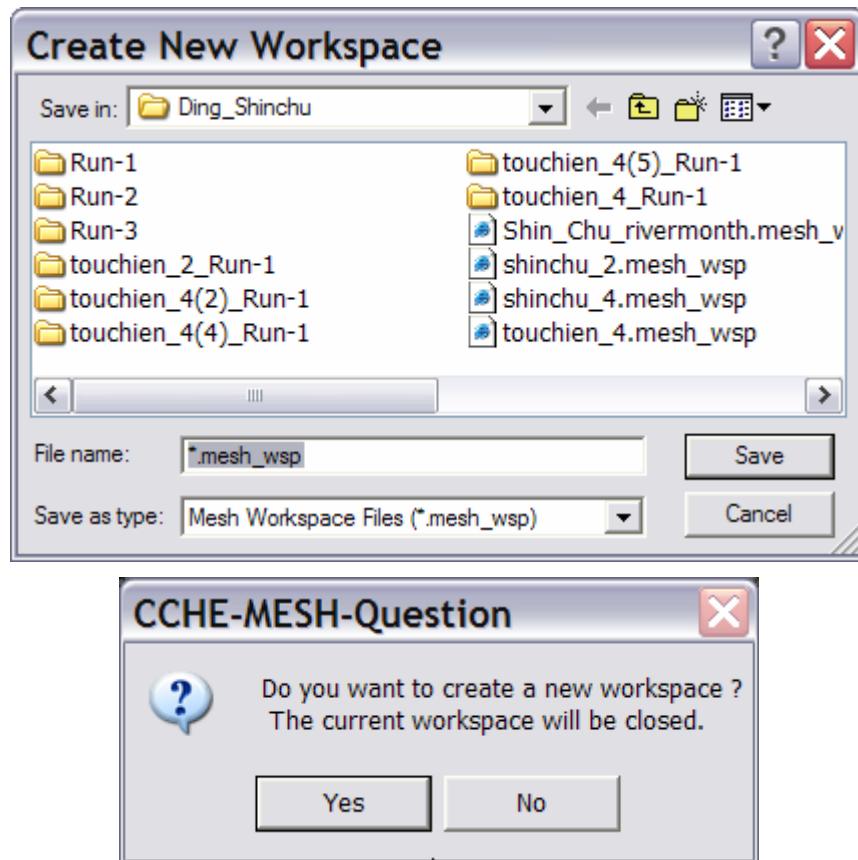


Figure 4-9

2. Open an existing workspace by selecting **Open Workspace File...** from **File** menu.

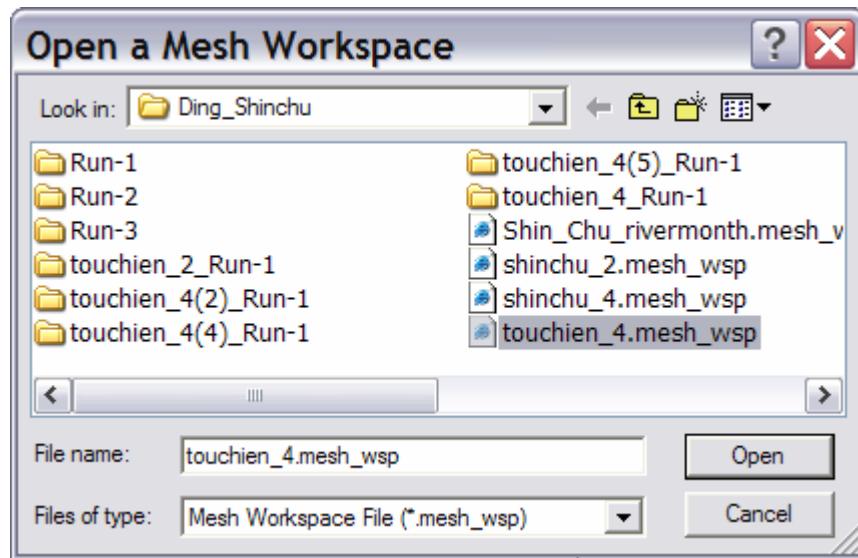
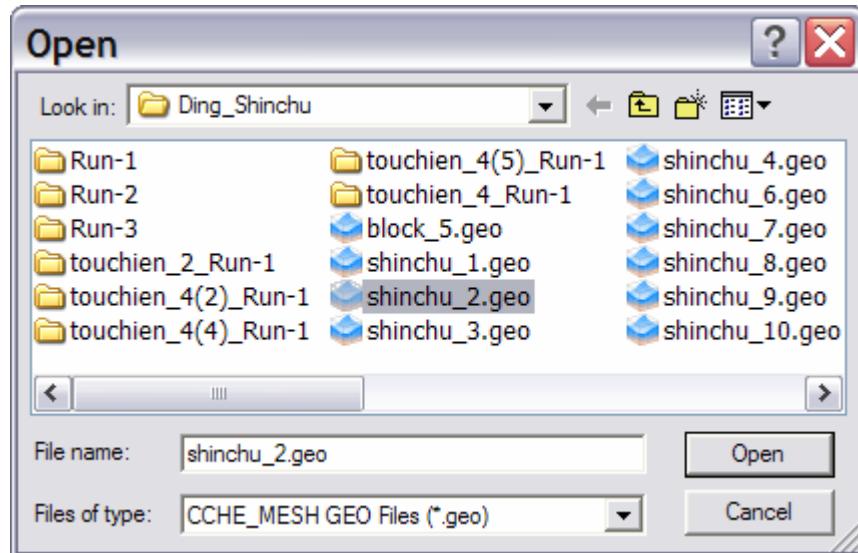


Figure 4-10

3. Directly load the supported file by selecting **Open Geometry File**, **Open Boundary File**, or **Import** popup menu from **File** menu. In this case, a workspace with the same name of the first file you loaded will be searched, then the user will be asked to confirm to load the associated workspace..



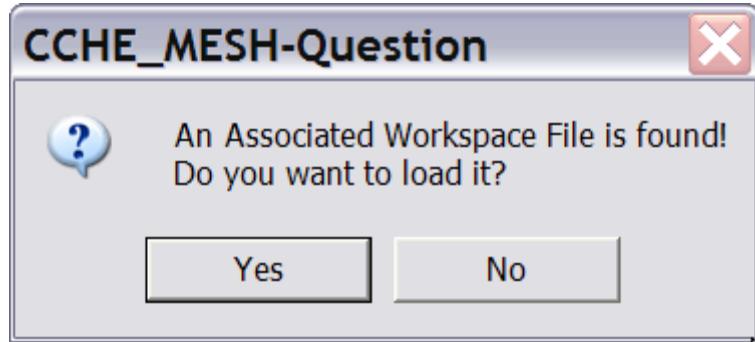


Figure 4-11

4.3 Define Blocks

To generate a mesh, the first step is to define the block with the control boundary that encloses the interested domain. For complex domains, you may need to divide the whole domain into multiple blocks with simpler geometry.

The control boundary is a closed curve represented by the control points. The CCHE-MESH is able to generate mesh based on an existing block boundary file (*.mesh_mb), a topography image (*.bmp), a topography database (*.mesh_xyz), a measured cross section database (*.mesh_mcs), a digital elevation model (ASCII) (*.asc), and the shape files (*.shp).

To define blocks, first you need to load one or more such files (*.bmp, *.mesh_xyz, *.mesh_mcs, or *.asc) as mentioned previously; and then you can use the **Block Editing** tools to create or edit the blocks.

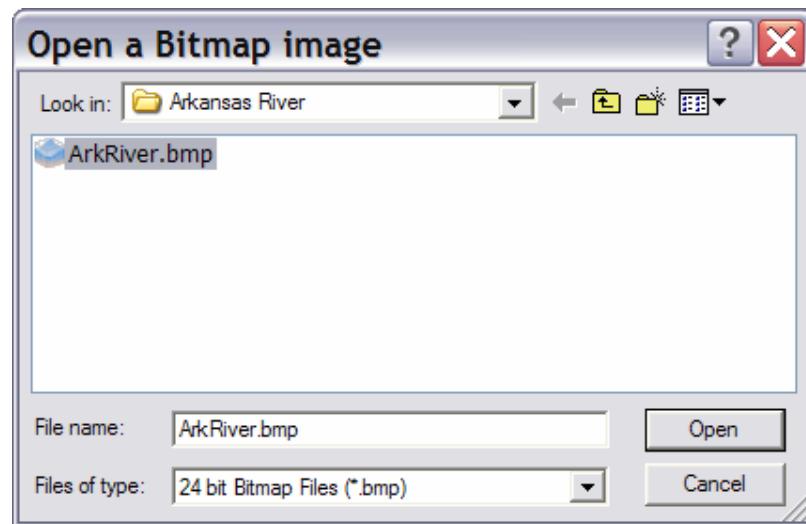
4.3.1 Import a Topography Image

A digitized topography map usually contains the topology information. The CCHE-MESH can create boundary from a topography map. However, the mesh generator can only identify 24-bit Bitmap image. If the map is of other type, you need to convert it to 24-bit Bitmap use some software.

4.3.1.1 Load Topography Image

- Select **Bitmap Image** in submenu **Import** from **File** menu.

- **Only 24-bit Bitmap image can be loaded**, otherwise an error message will be popped up.



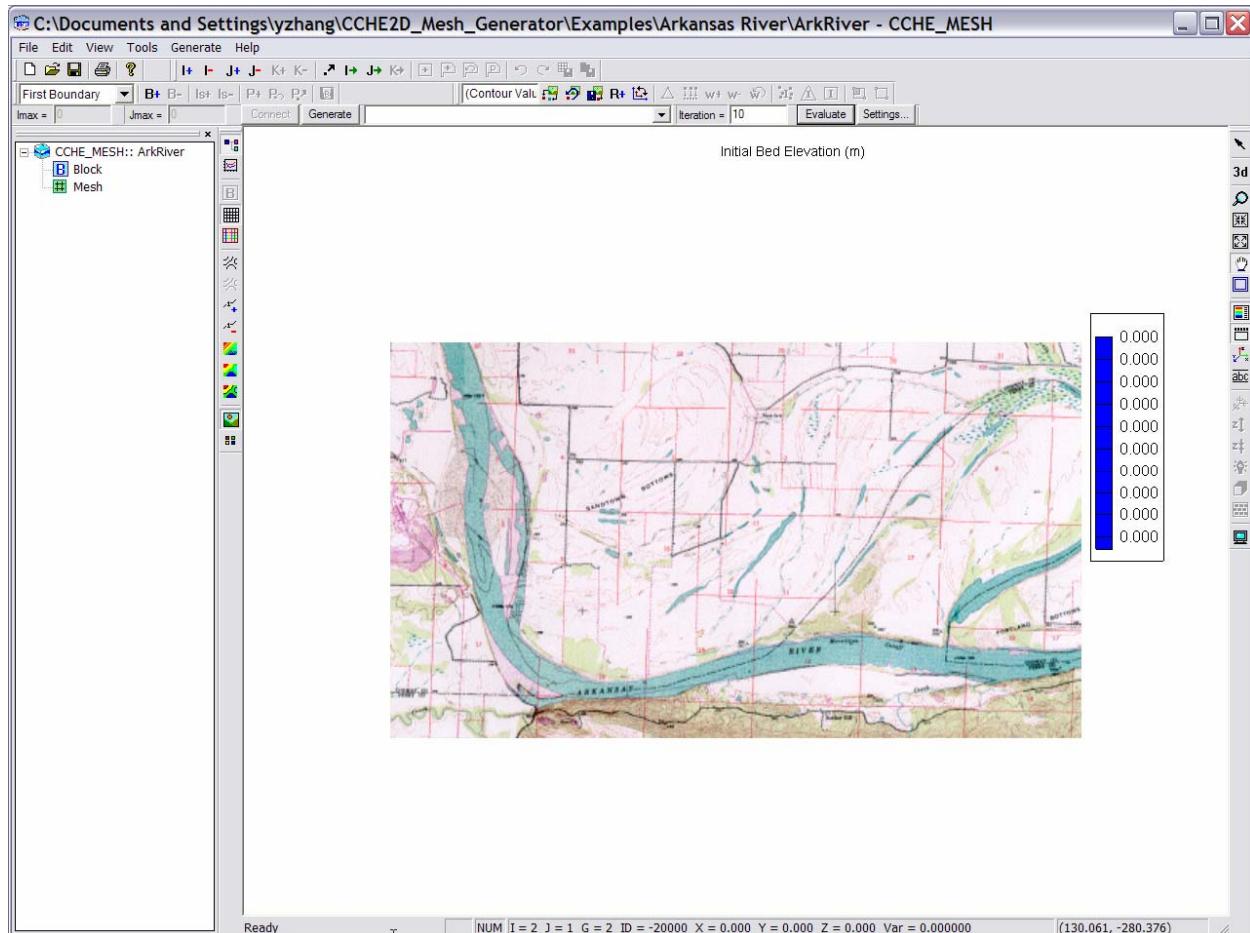


Figure 4-12

4.3.1.2 Coordinates Transformation

- Once the image is loaded, if the map is not geometrically referenced map that has a file (*.grm) contains the coordinates transformation information, you need to do the coordinate transformation by defining two reference points on the image.
- Select **R+** from **Image Editing Tools** toolbar or **Define Reference Point** in **Topography Image** submenu from **Tools** menu, and then click the point which you have the reference coordinates information on the image.

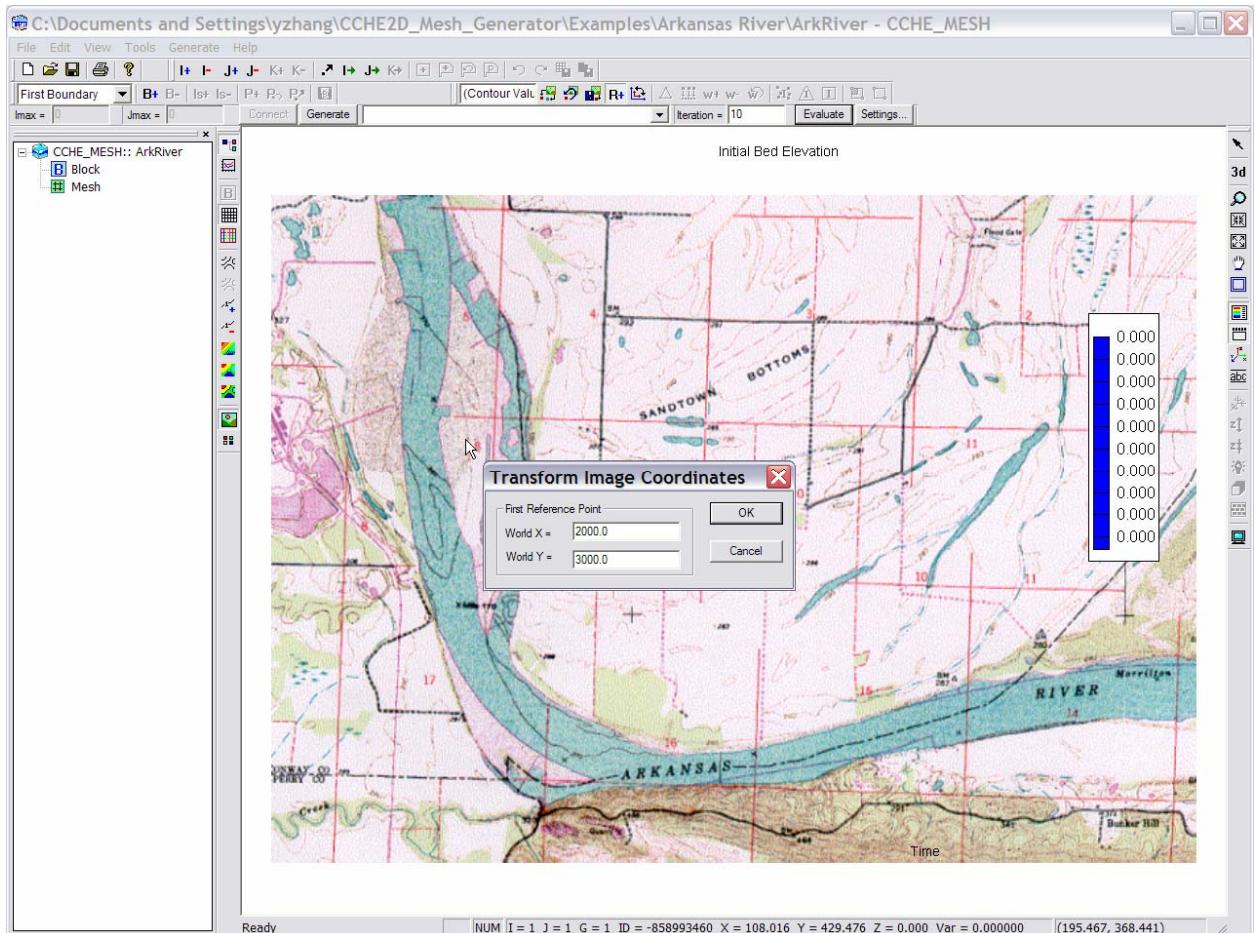


Figure 4-13

- Repeat to define the second the reference point.

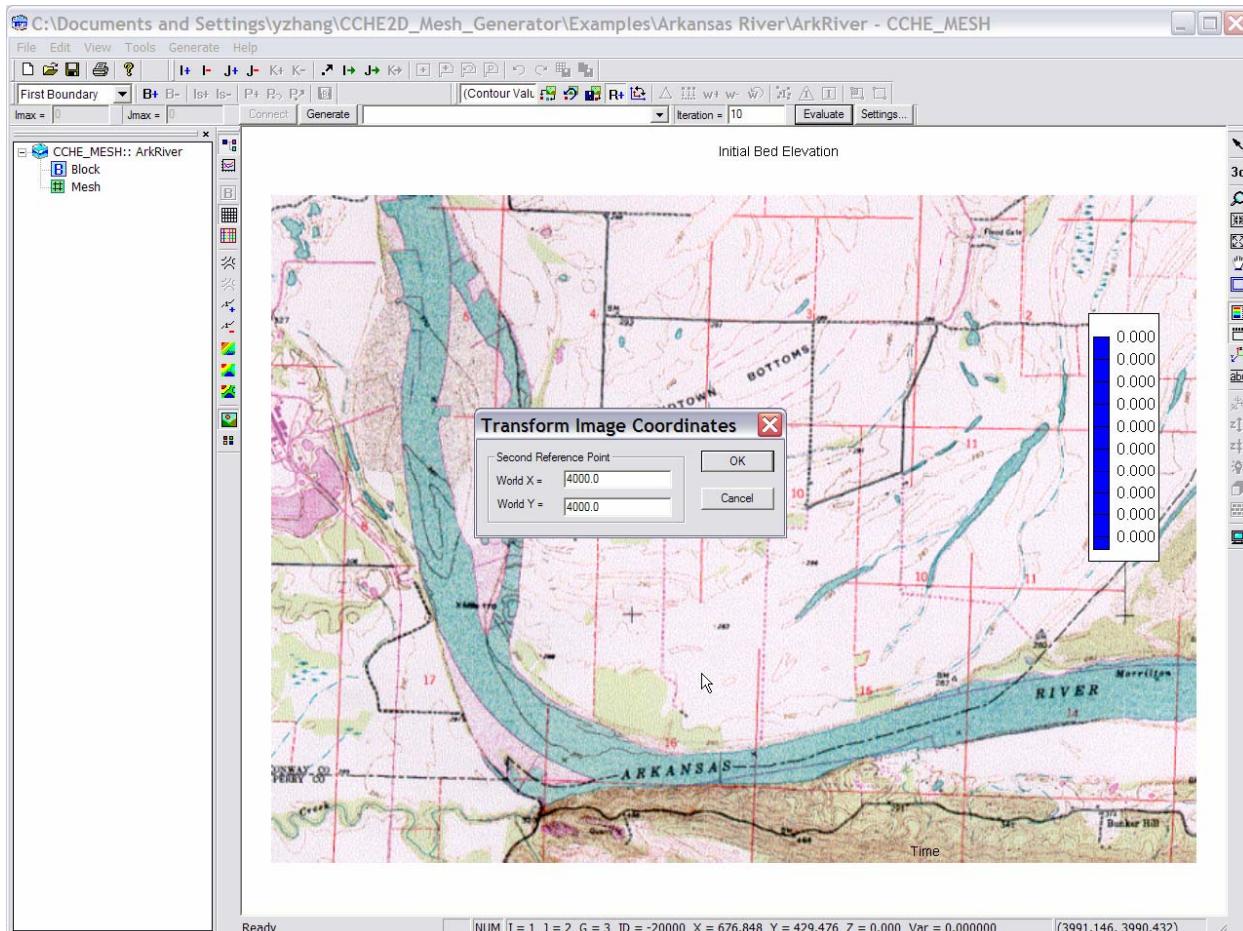


Figure 4-14

- Once the transformation is completed, the transformation information will be saved into a file (*.grm) with the same name as the image file. Next time when loading this image again, the coordinates transformation information (*.grm) will be loaded automatically and you don't need to do the transformation again.

4.3.1.3 Digitize Image

If the topography image contains the topology contour information, the CCHE-MESH is capable of extracting those contour lines from the image into a topography database. You can use the image tools from **Tools** menu or toolbar to do the digitization.



To digitize the image,

- Select to enter the digitization status.
- Then select a contour line by inputting the value of this contour into **(Contour Value)**.
- Add points along the selected contour lines.
- Repeat the operations on the other contour lines.

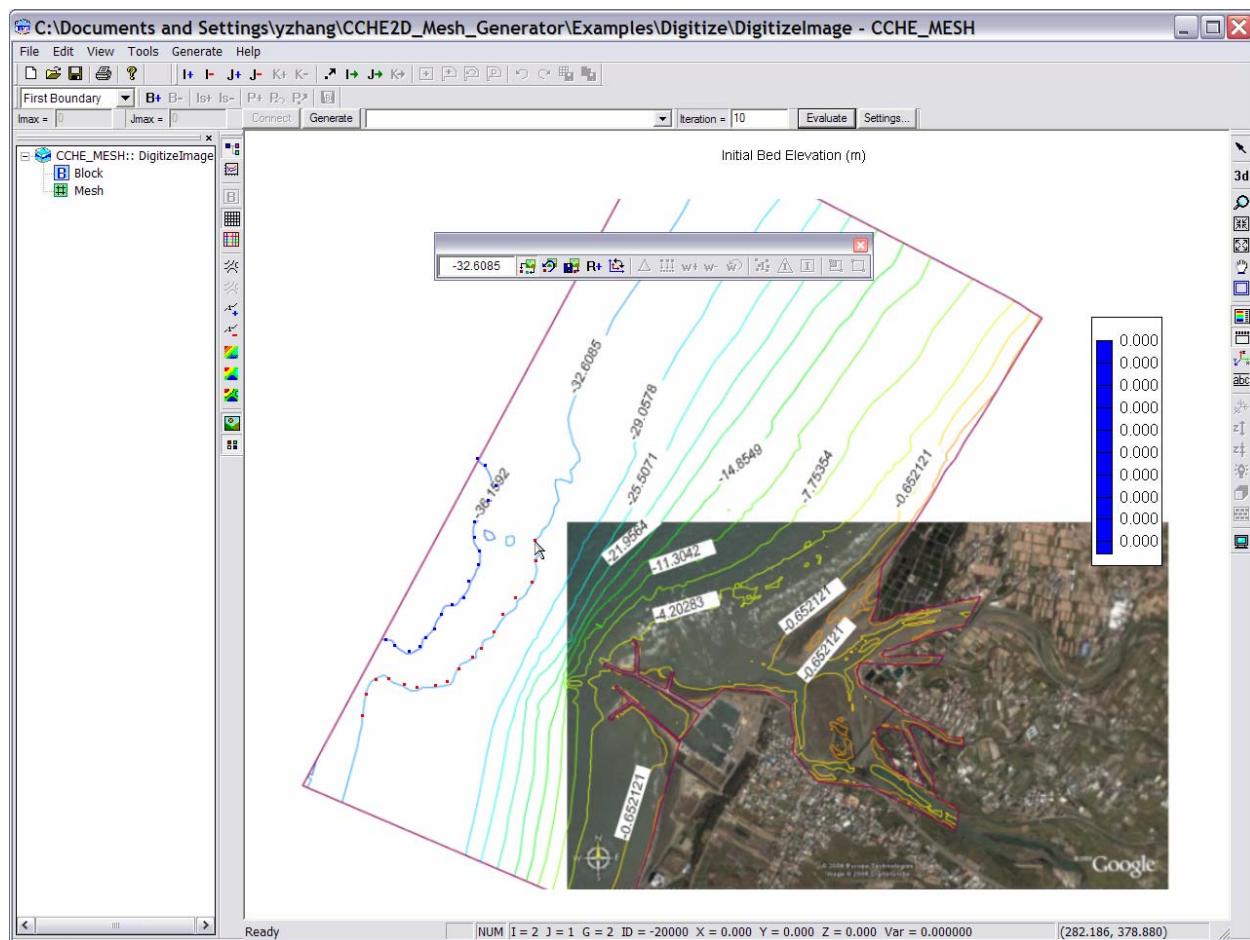


Figure 4-15

During the digitization process,

- You can undo the previous point anytime by selecting .
- It is highly recommended to save the digitized information from time to time by selecting .

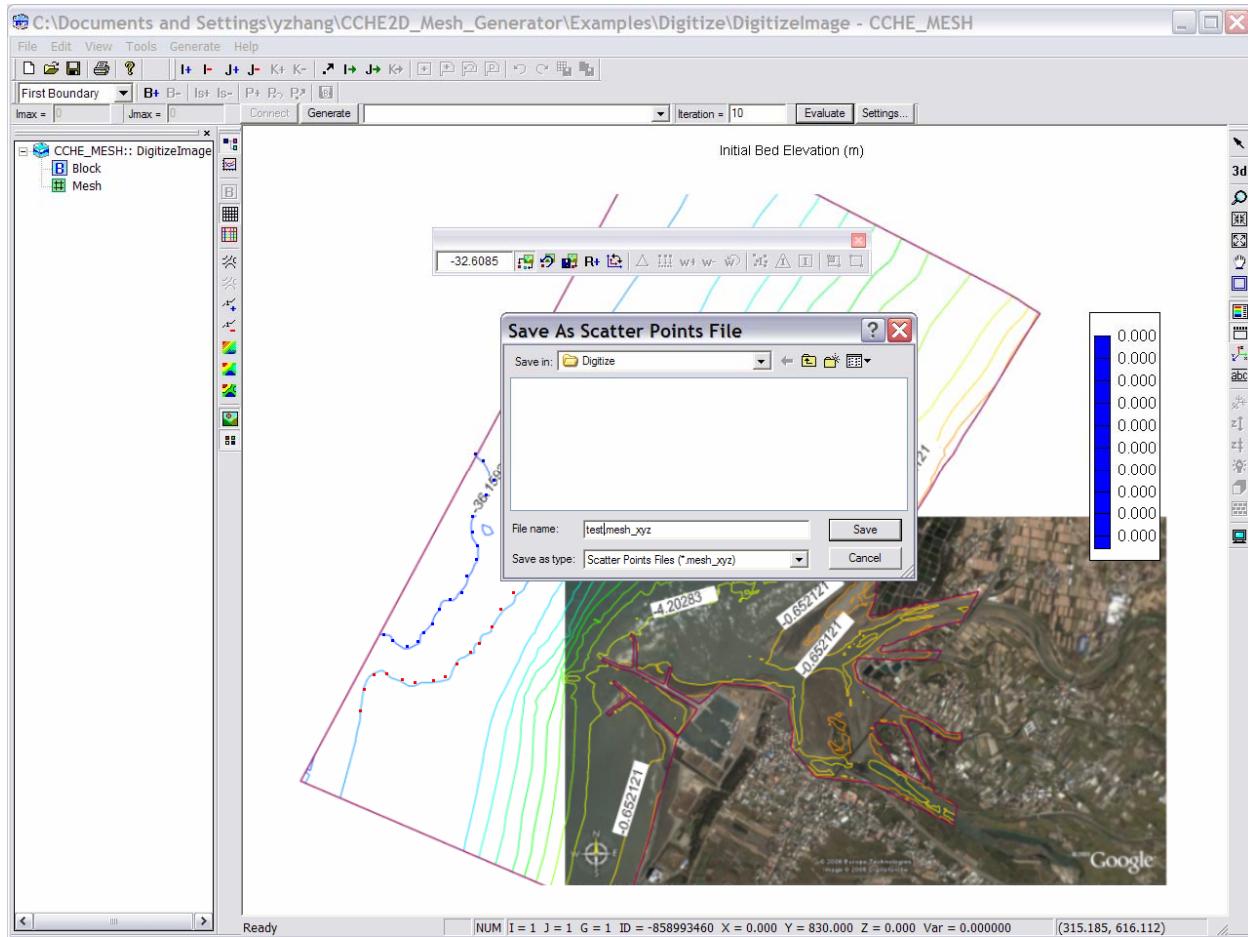
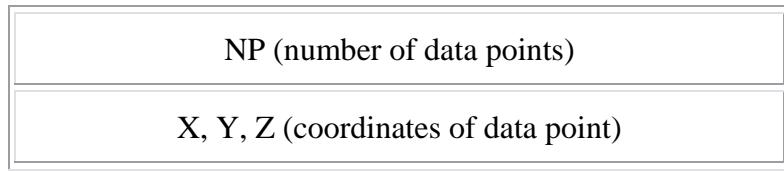


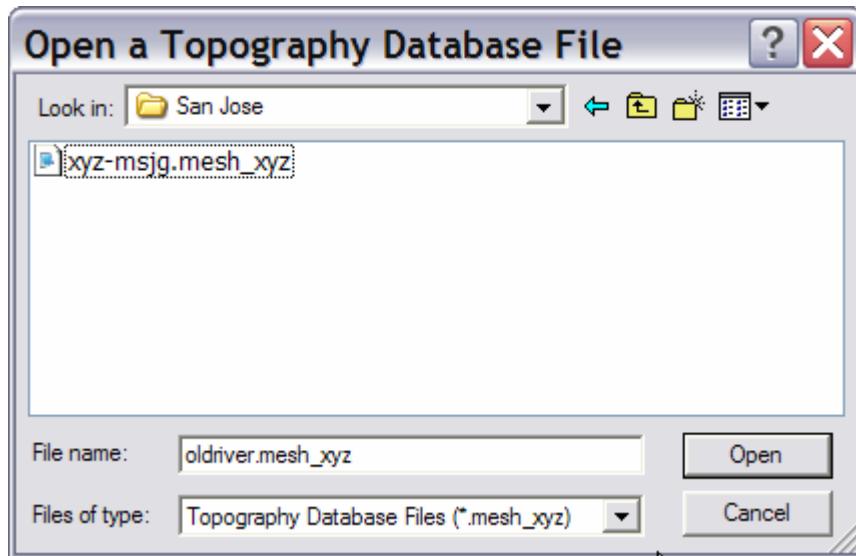
Figure 4-16

4.3.2 Import a Topography Database

In the CCHE-MESH, a file with an extension of “mesh_xyz” is identified as the topography database file. As shown as follows, the format of this file is quite simple.



You need to select **Topography Database (*.mesh_xyz)** from **Import** submenu in **File** menu to load a topography database.



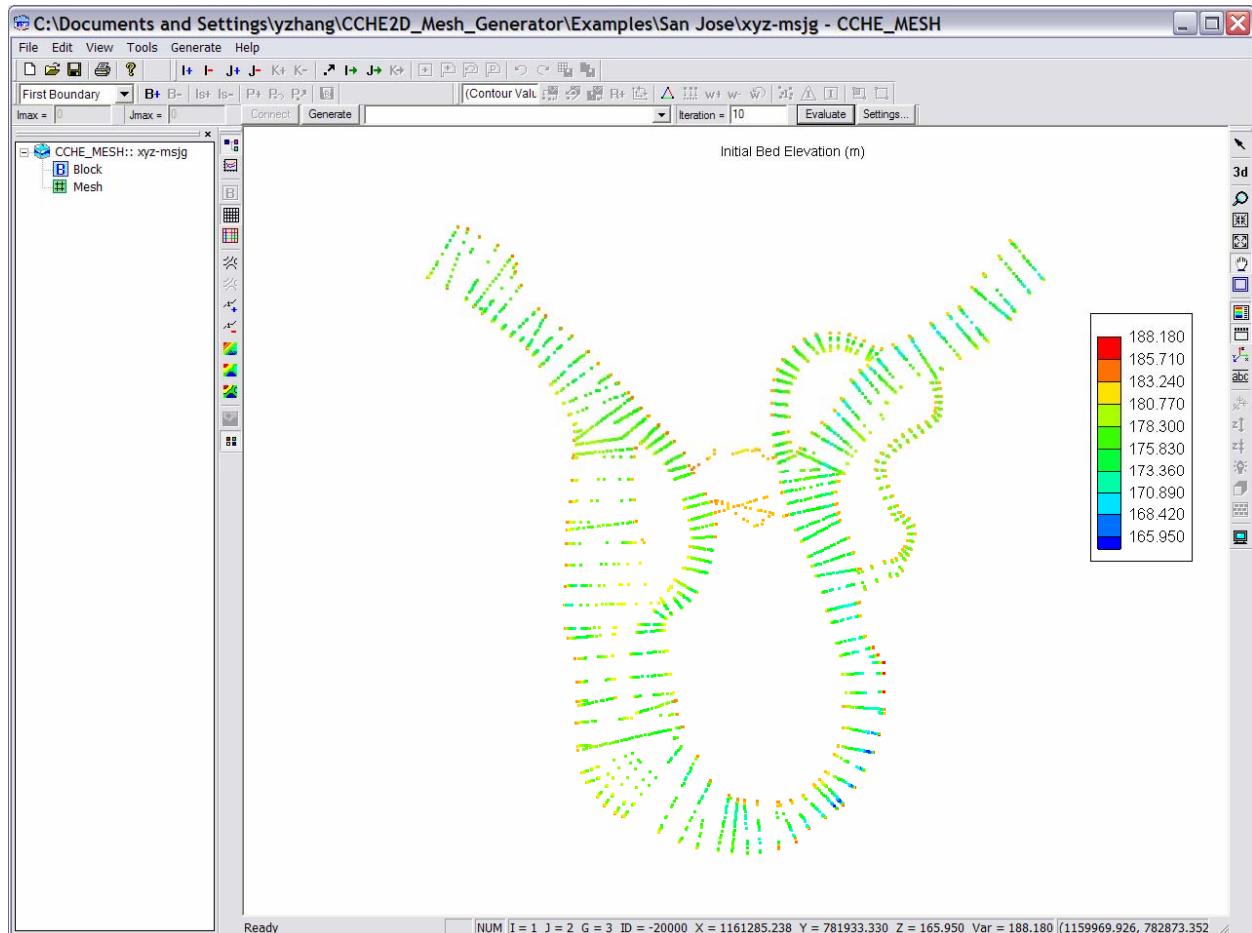


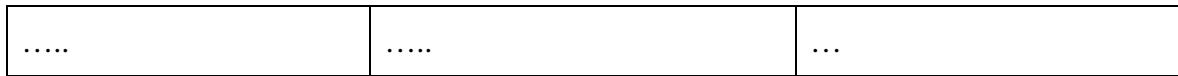
Figure 4-17

4.3.3 Import Measured Cross Sections

The measured cross sections file (*.mesh_mcs) is also a kind of topography database. Its difference from the topography database file (*.mesh_xyz) lies in that the points are organized cross section by cross section.

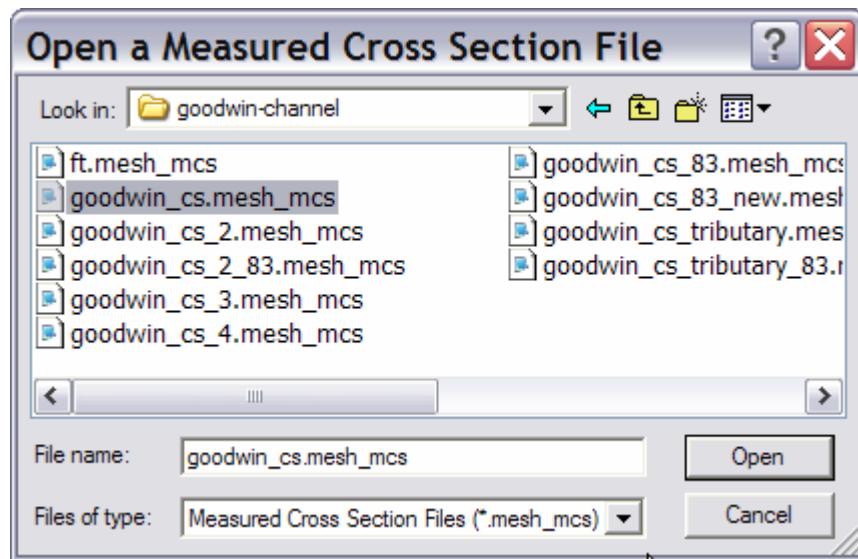
The format of the measured cross sections file is listed as follows.

Number of Cross Sections (CS)		
CS Number	Number of Points	Direction Flag (Optional)
X	Y	Z



4.3.3.1 Load Measured Cross Sections

You need to select **Measured Cross Sections (*.mesh_mcs)** from **Import** submenu in **File** menu to load the measured cross sections file.



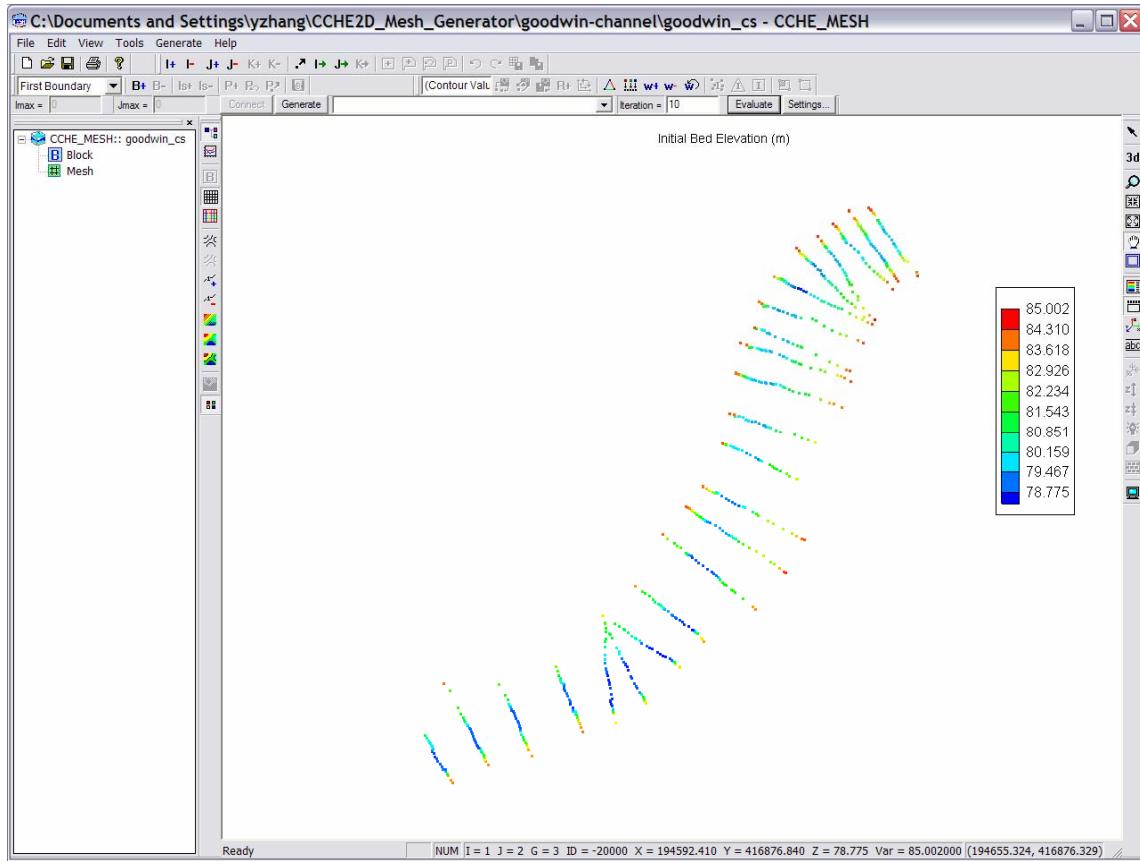


Figure 4-18

4.3.3.2 Refine Measured Cross Sections

In case that the measured cross sections are sparse which will affect the accuracy of the bed interpolation, the CCHE-MESH provides you the **Topography Database** tools to refine the original cross sections.



There are three kinds of refinements.

- **Direct refinement without auxiliary wet lines.** You just simply select **Refine Topography Database** from **Topography Database** submenu in **Tools** menu or from **Tools** toolbar. The refined measured cross sections will be saved into a topography database file (*.mesh_xyz).

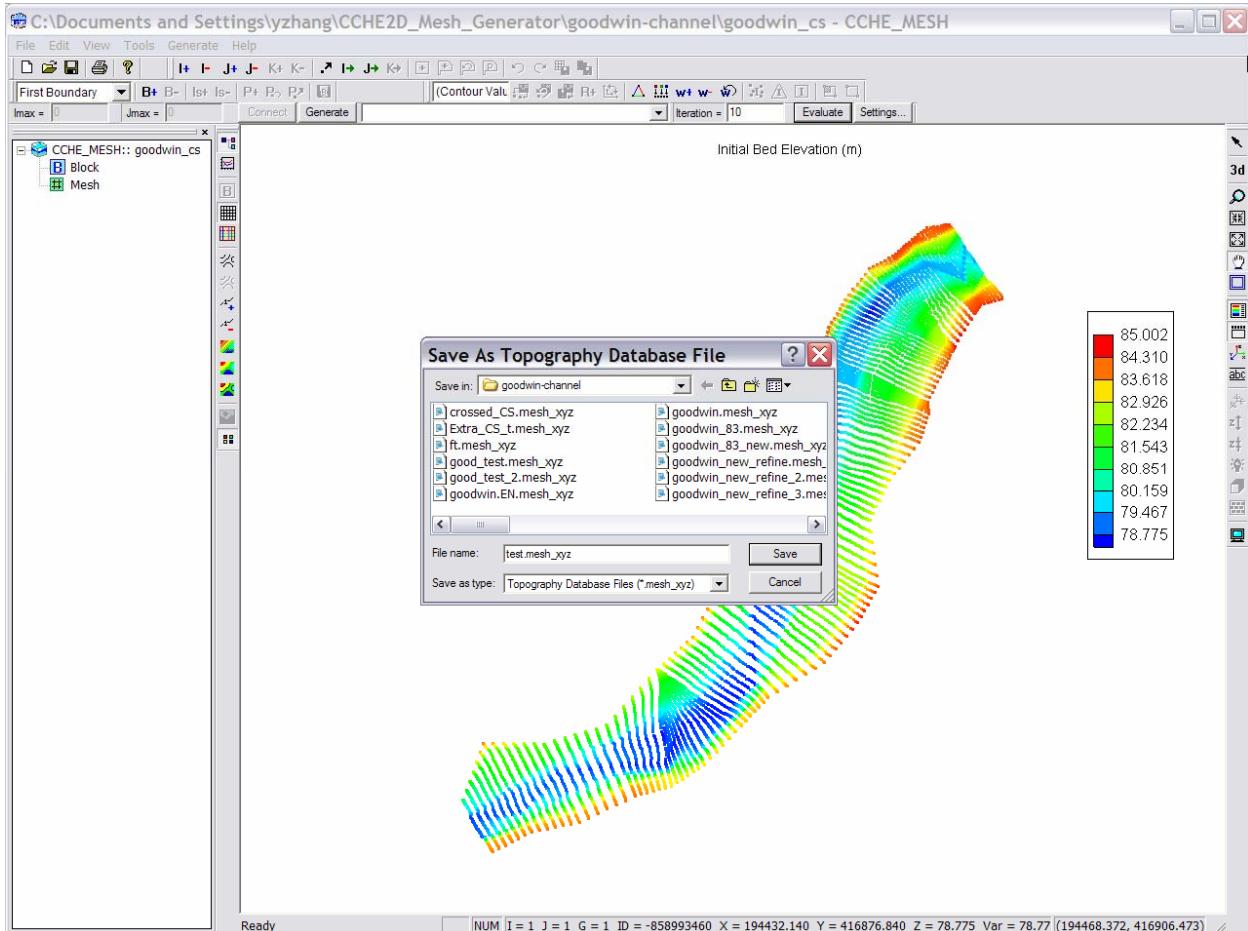
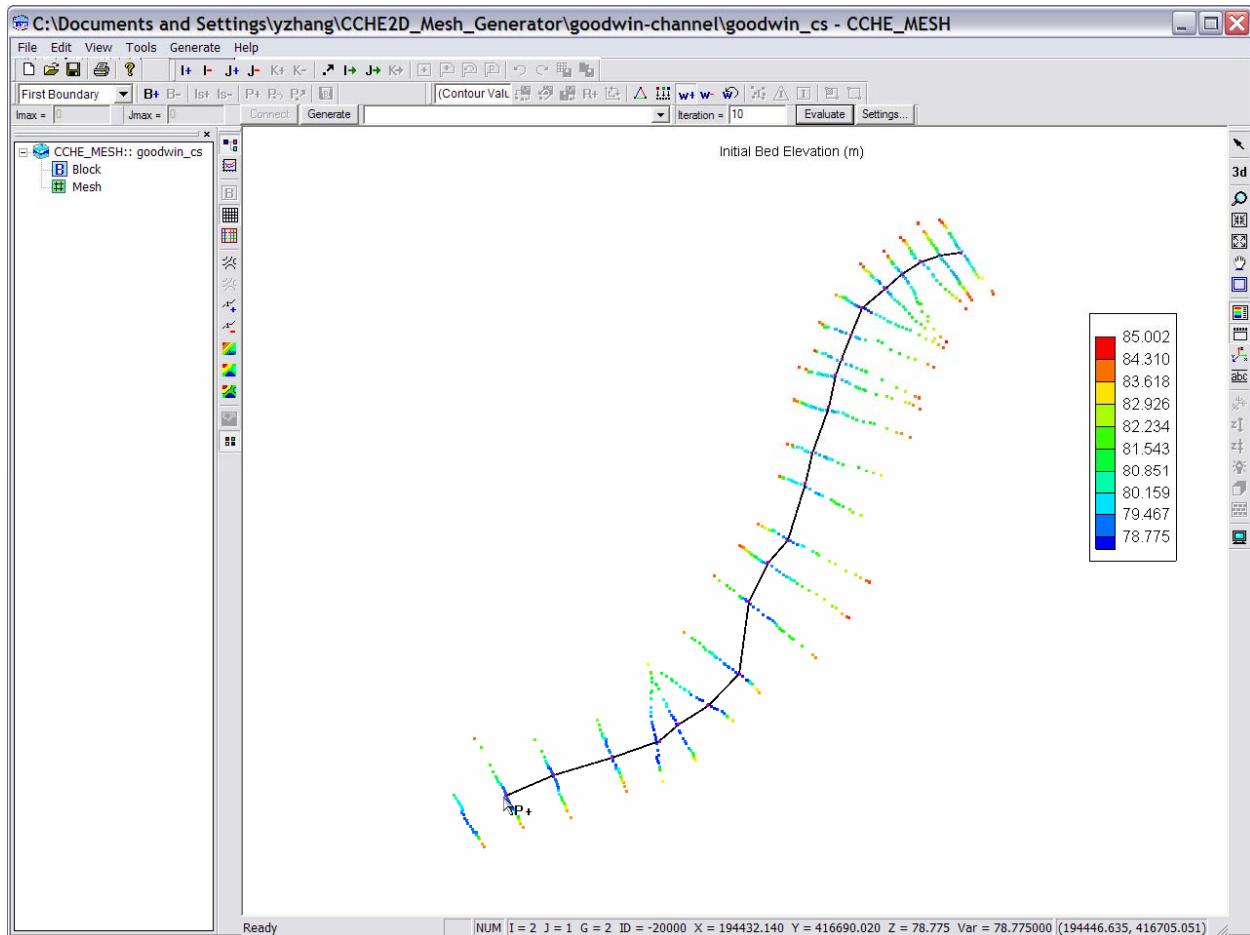


Figure 4-19

- **Refinement with one auxiliary wet line** which defines the refinement direction. Since the linear interpolation is used between cross sections, the wet line will help to guide the refinement to reflect the real topology as accurately as possible. To define a wet line, select **w+** and then add points to this line by clicking the desired location for each cross section. **NOTE: the number of points of the wet line must be equal to the number of the cross sections.** After defining a wet line, you can select **---** to refine the cross sections. When defining the wet line, you can undo the previous point by selecting **w-**. You can also delete a wet line by selecting **w-**.



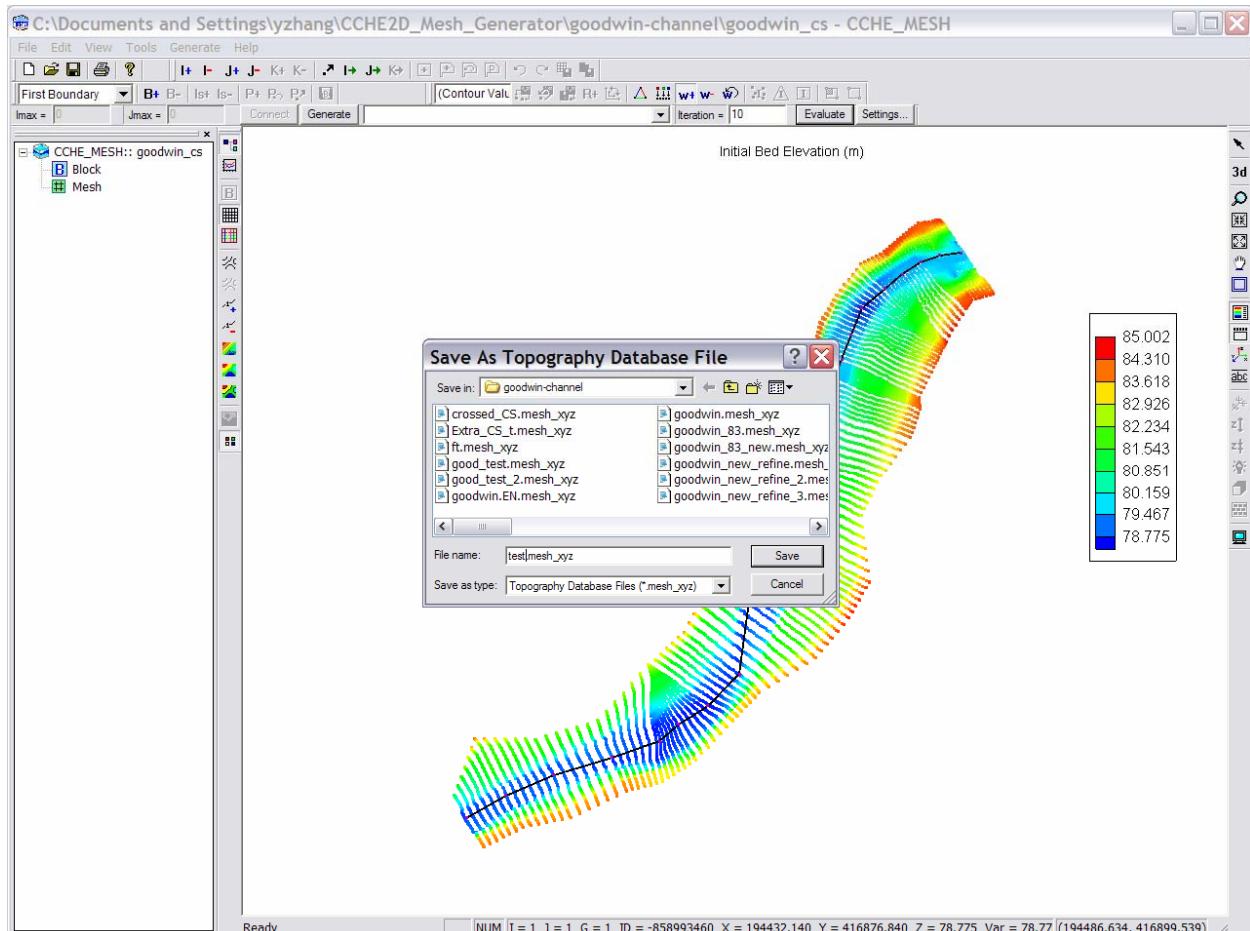


Figure 4-20

- **Refinement with two auxiliary wet lines.** You are allowed to define maximum two wet lines.

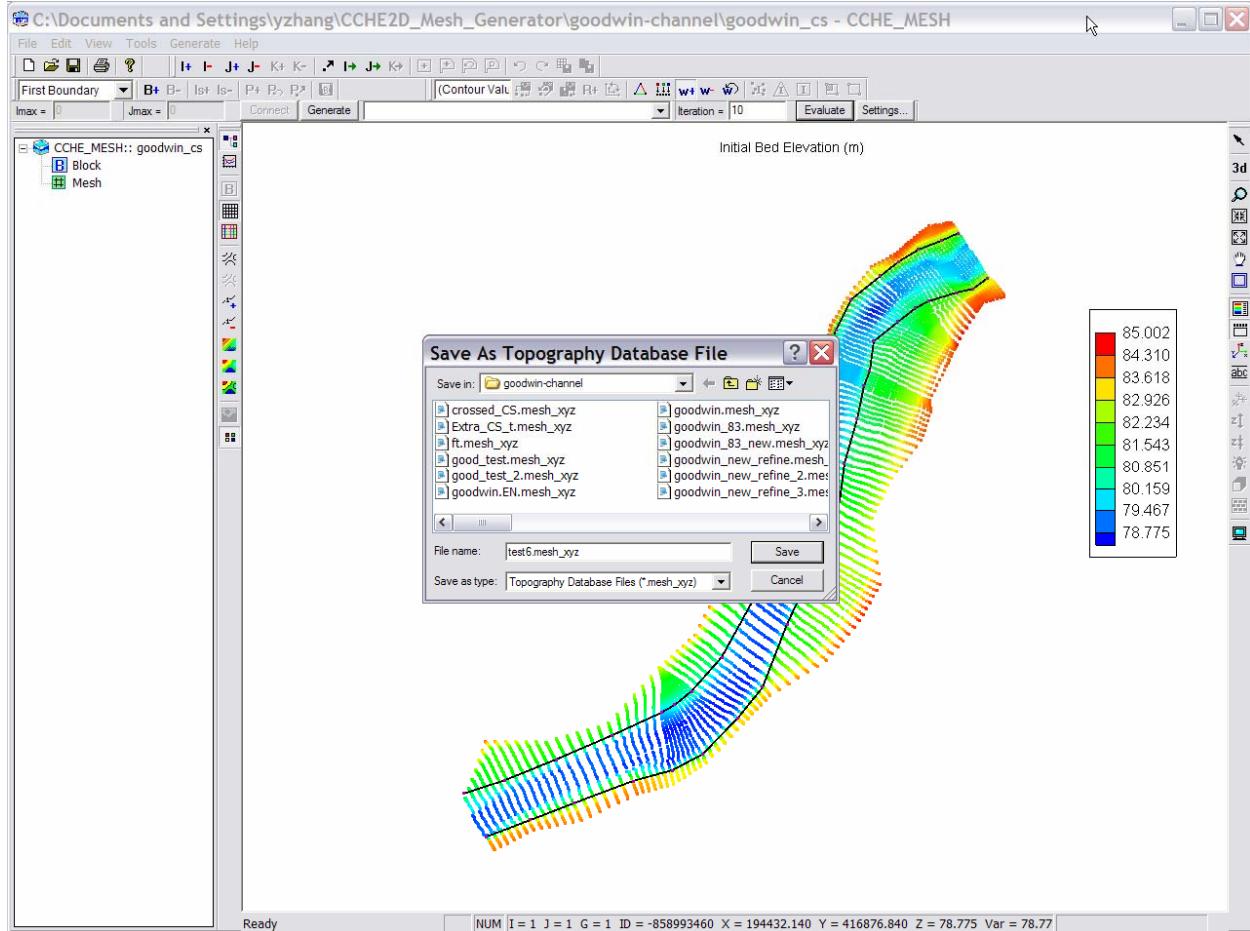


Figure 4-21

4.3.4 Import Digital Elevation Model

The Digital Elevation Model (DEM) file is a kind of popular topography database. The CCHE-MESH can only identify the ASCII coded DEM file and the users need to convert the format of the original DEM file using other software, i.e., ArcGIS.

The following table lists the format of the ASCII code DEM file.

Number of Columns
Number of Rows
X Coordinate of the South-west Corner Point

Y Coordinate of the South-west Corner Point
Size of the Element
Value of the No-Data Point
Elevation by Row

You need to select **Digital Elevation Model (ASCII) (*.asc)** from **Import** submenu in **File** menu to load the DEM file.

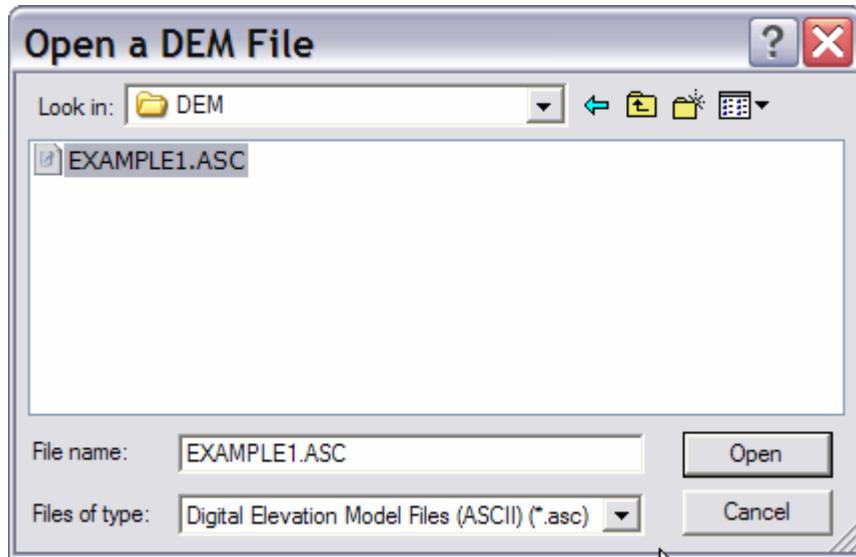


Figure 4-22

In the **Load Digital Elevation Model (ASCII)** dialog window, the header information is listed and you can select actions operated on this DEM file.

- **Convert Units:** you may need to convert the units from US system to SI system.
- **Generate Mesh:** you can also generate a mesh directly from this DEM. NOTE: the maximum number of I lines can not exceed the number of columns and the maximum of J lines should be less than the number of rows.

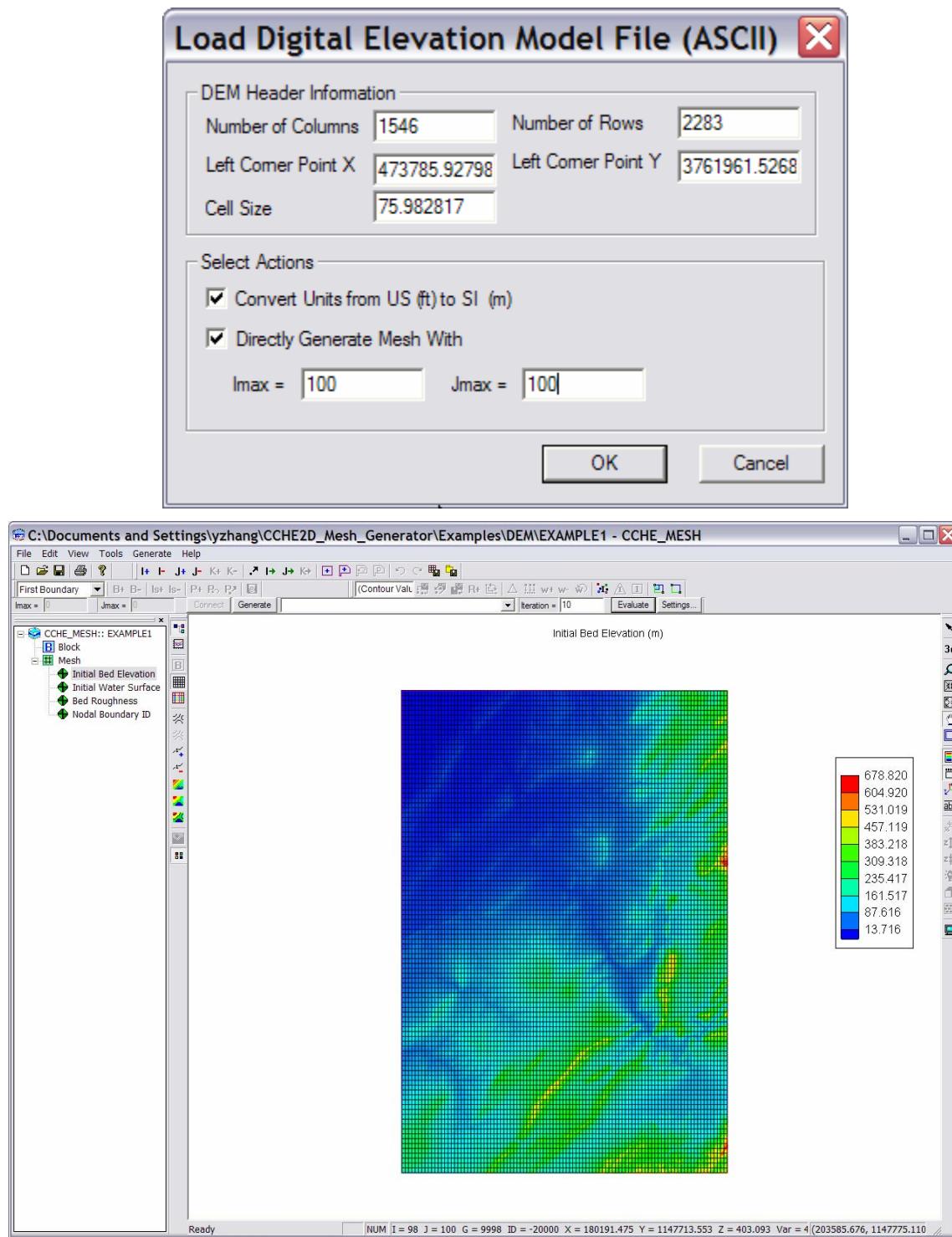


Figure 4-23

4.3.5 Import Shape File

The Shape files are GIS data files supported by ArcView and ArcGIS software. The CCHE-MESH can also import these files. To load one or multiple shape files, you need to select **Shape Files (*.shp)** from **Import** submenu in **File** menu.

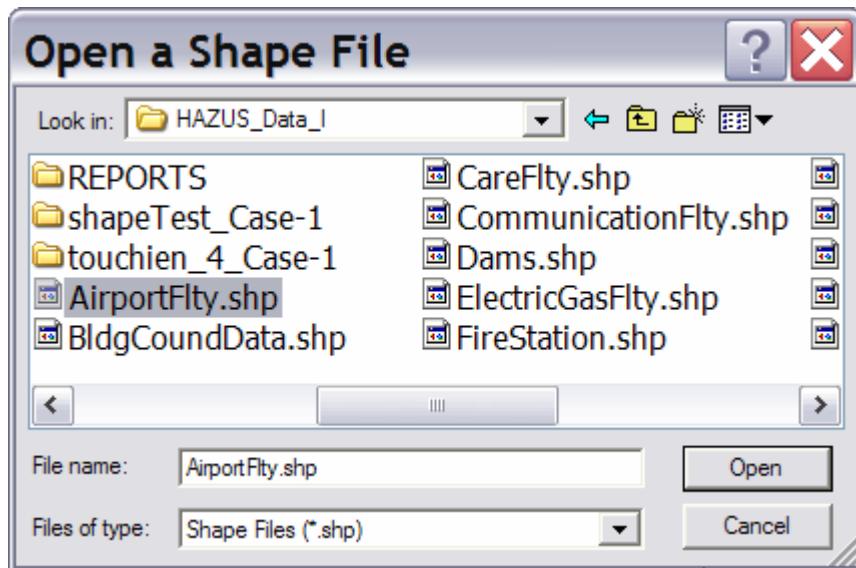


Figure 4-24

In the **Shape File Loader** window, all the shape files in the same directory as the shape file you selected will be listed. You can select the shape files you want to load. By default, all the shape files will be loaded.

- Click “>> All” to select the whole shape files.
- Select a file on the left panel “Found” and then click “>>” to add this selected file to the “To Load” list.
- Select a file on the right panel “To Load” and then click “X” to delete this selected file from the “To Load” list.
- Click “X All” to delete all files from the “To Load” list.

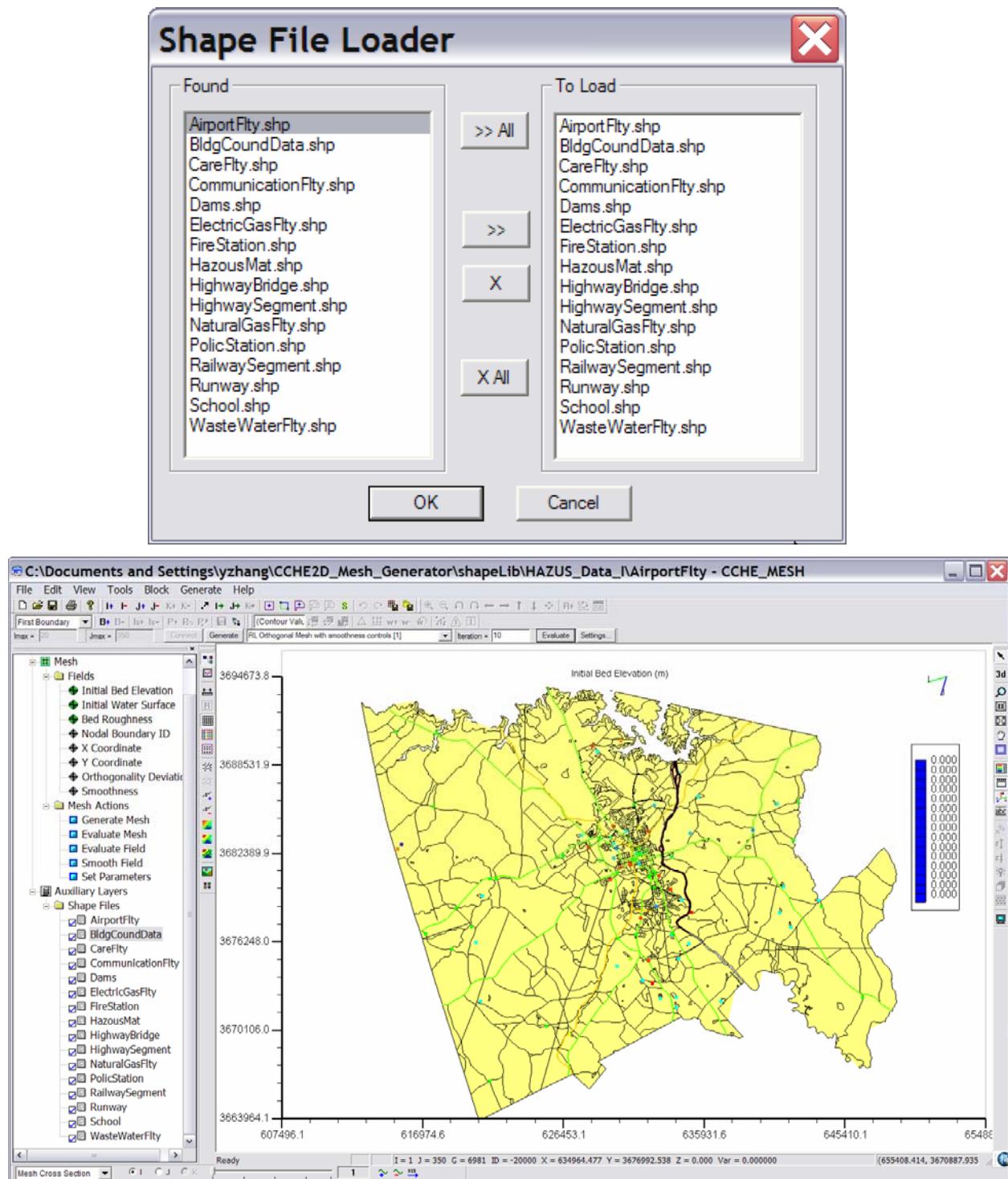
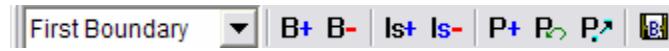


Figure 4-25

4.3.6 Define Block Boundary

After loading the supported files, such as topography image (*.bmp), topography database (*.mesh_xyz), measured cross sections (*.mesh_mcs), ASCII coded DEM (*.asc) files, and the shape files (*.shp), you can use the **Block Editing** tools to define the blocks.



For example, a topography database is loaded into the CCHE-MESH as shown in Figure 4-26.

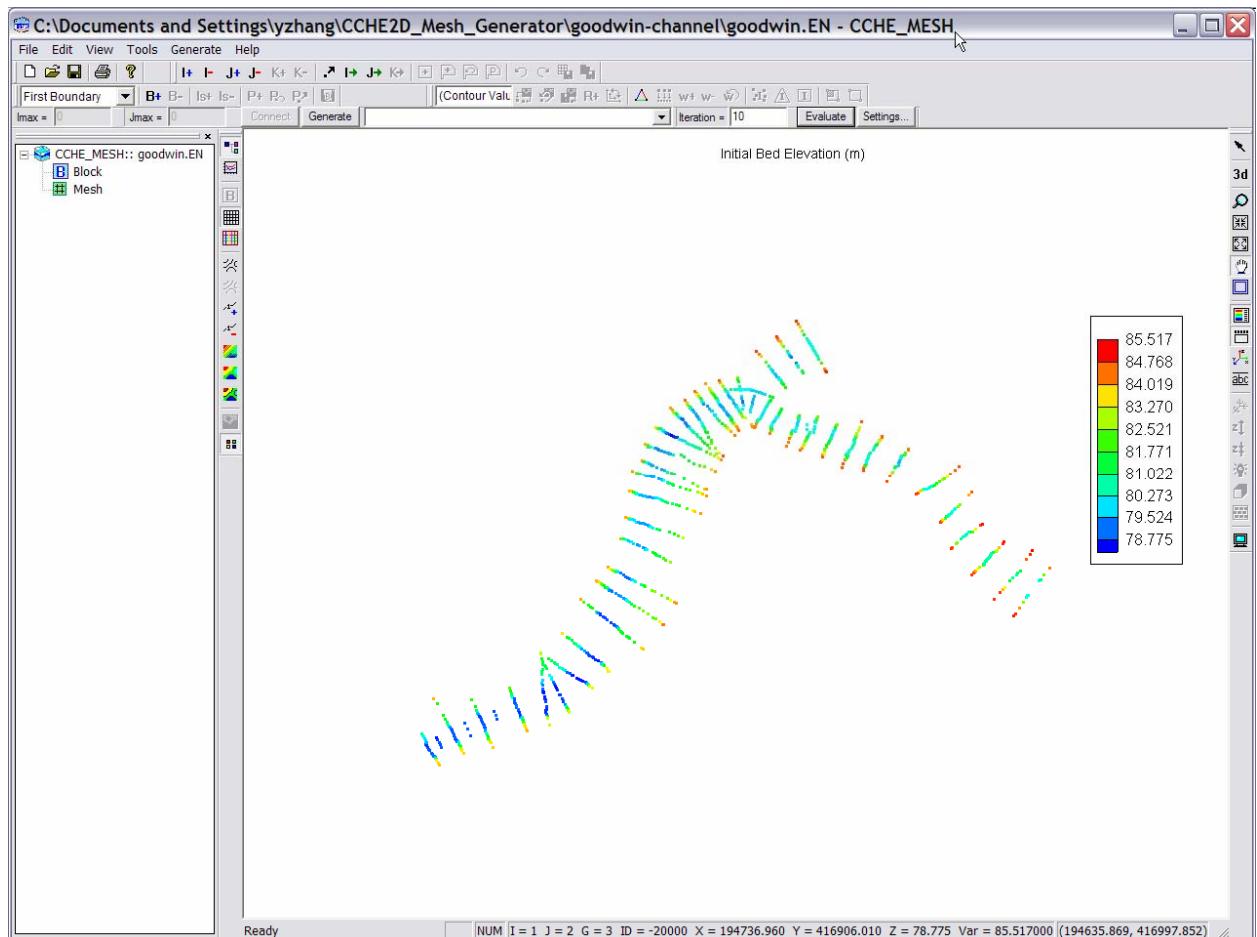


Figure 4-26

- To **define a new block**, select **B+** to enter the editing status. As stated in Chapter 2, the CCHE-MESH uses a two-boundary method to generate algebraic mesh, so a block is represented by two boundaries. For the next step, you need to define two boundaries for this new block.
- To **define the first boundary** for the current block, first select “**First Boundary**” from **Bnd Type** selector **First Boundary** **▼**, and then left click the desired place on the plot to add a boundary control point to this boundary. Repeat adding the boundary control points until all the control points along the first boundary are identified.

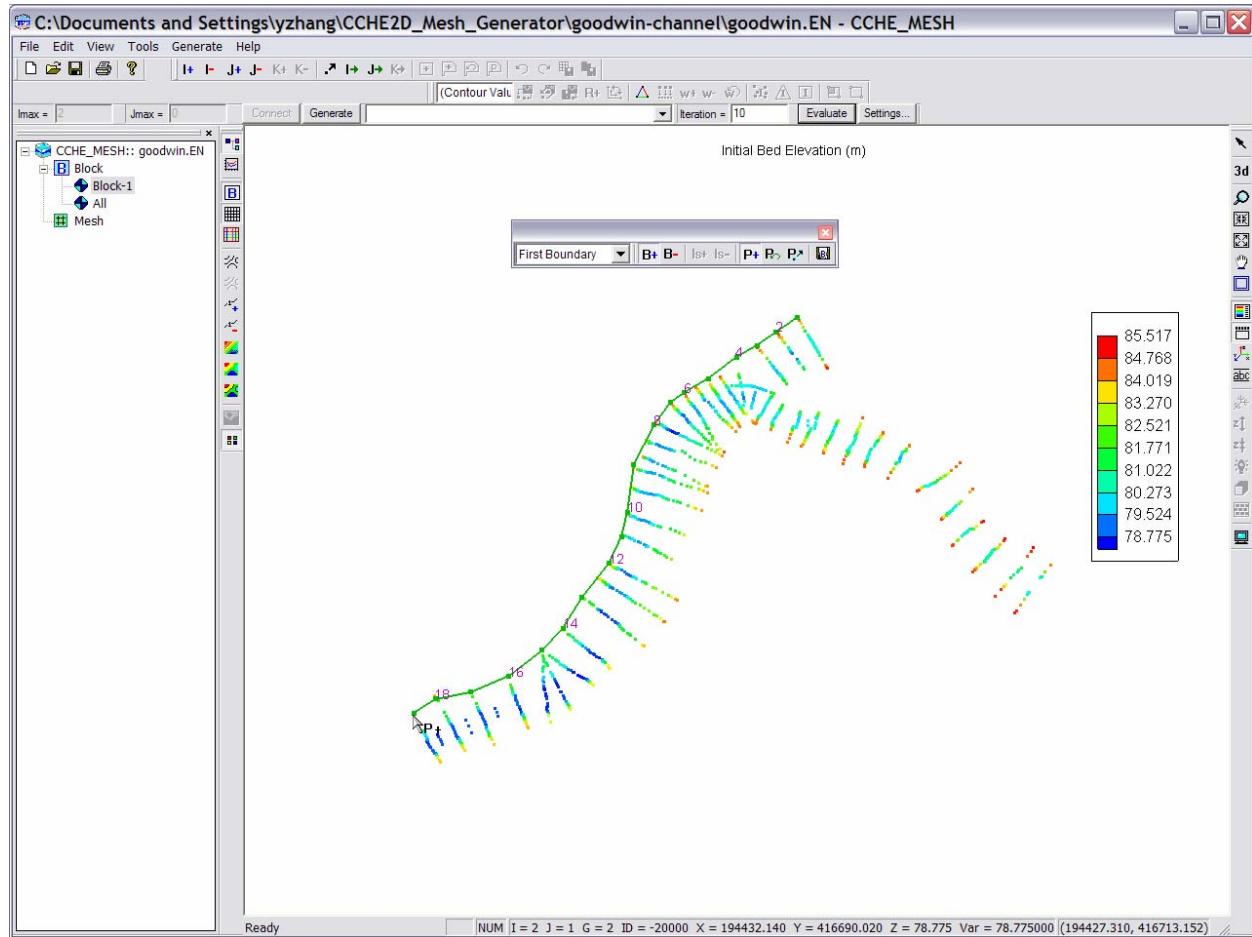


Figure 4-27

- To define second boundary, select “Second Boundary” from the Bnd Type selector, then follow the same procedure as defining the first boundary.

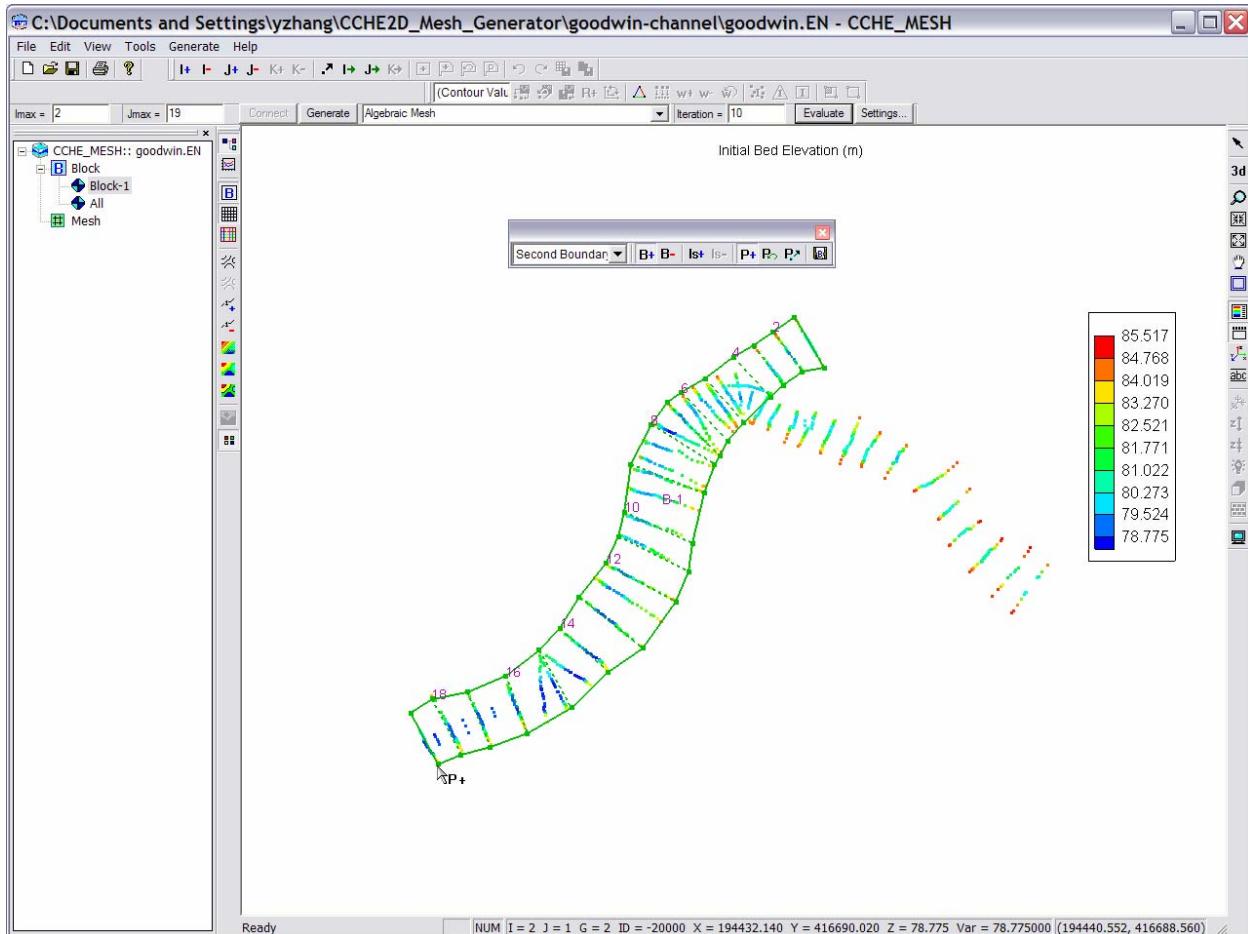


Figure 4-28

- Tips:** You can **undo the previous action** anytime when defining first or second boundary by clicking ; and you can also move the boundary control point by selecting first and the move the selected point to the desired place.
- You can **add/define island** for the current block. An island is a kind of special “block” which is also represented by two boundaries. To **add island** to the current block,

- Select **Is+** to **add a new island**, and then define two boundaries following the same procedure as defining two boundaries for a block.

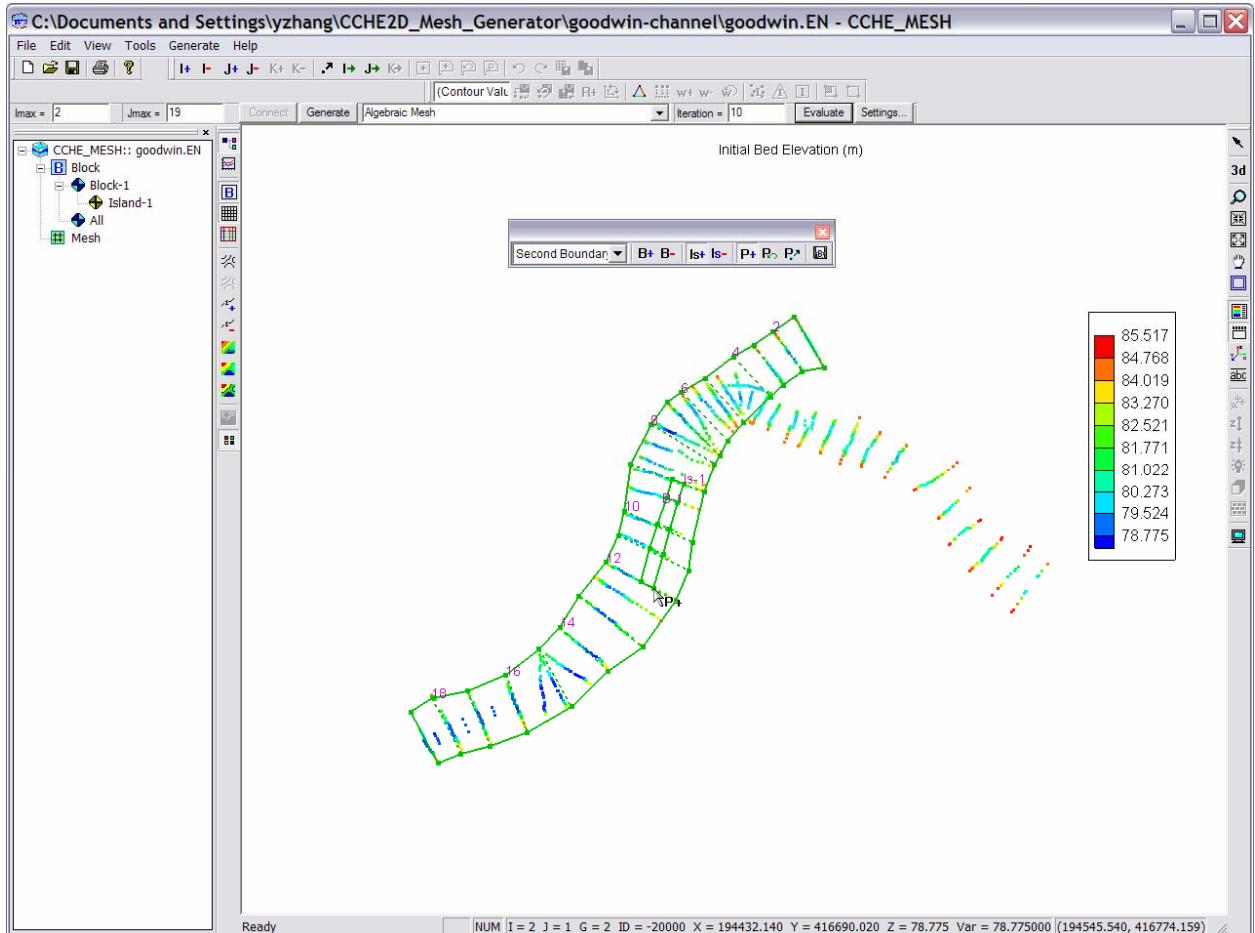


Figure 4-29

- To **add more islands**, repeat the above step.
- Note:** The relative location of the two boundaries of the island must be the same as that of the current block.
- Note:** The number of control points of the island should be less than the number of the control points of the current block.
- Select **B+** to **add second block**, and then define two boundaries for the block-2.

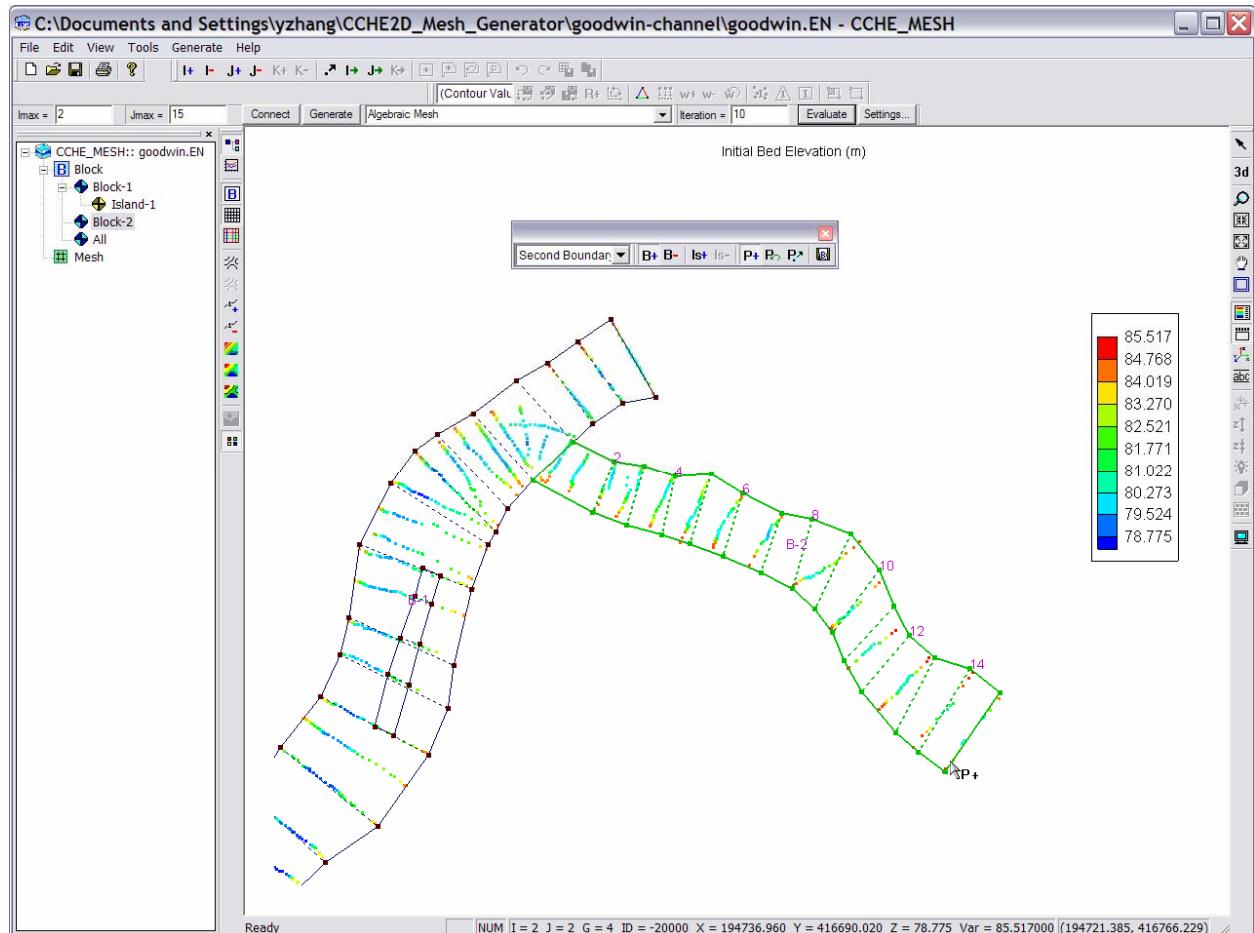


Figure 4-30

- To add more blocks, repeat the above steps. Notes:
 - The added block must be connected with the existing blocks.
 - Only one interface exists between two connected blocks.
 - A blue line will be shown to indicate an interface.
 - There are at least two points for one interface.

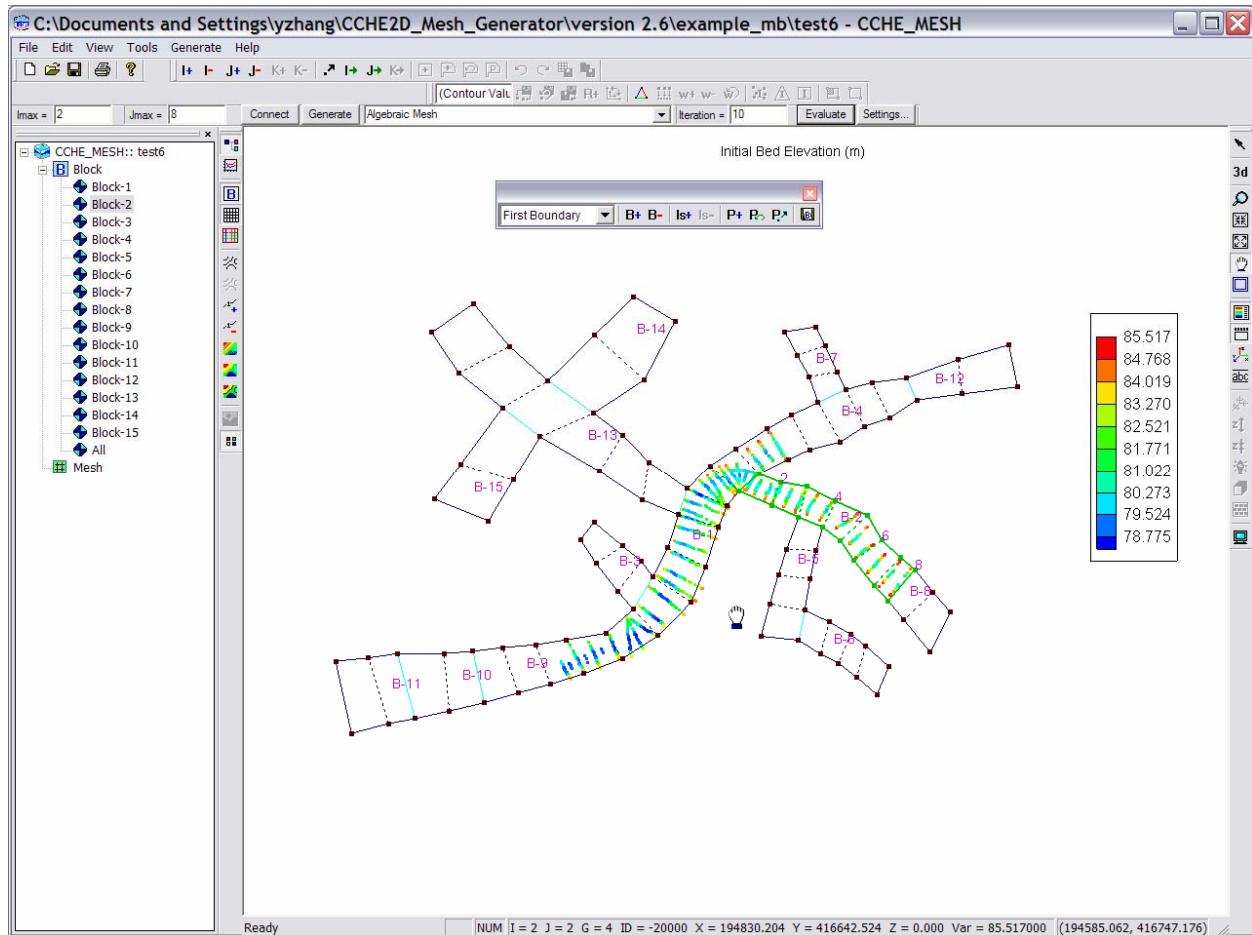


Figure 4-31

- To **save** the defined blocks, select to save it into a mesh_mb file. It is recommended to save the block from time to time when defining blocks. The CCHE-MESH allows you to save the unfinished blocks.

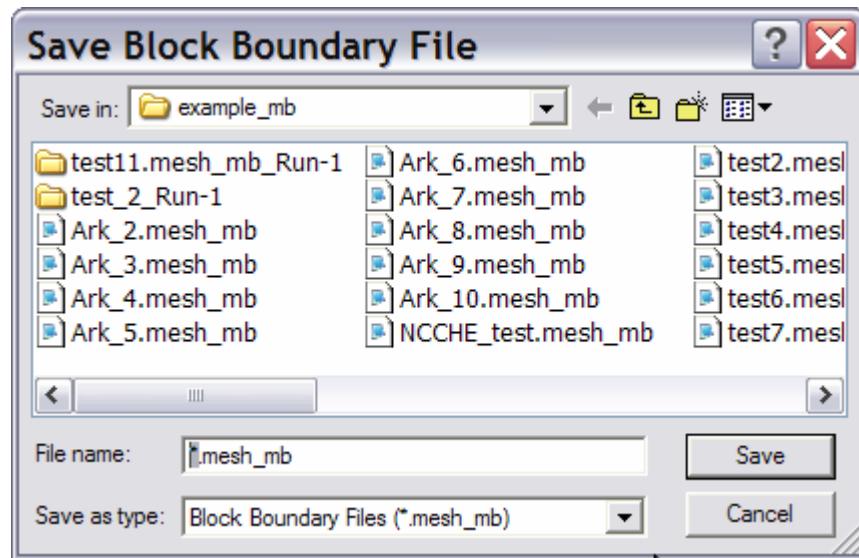


Figure 4-32

4.3.7 Set Block Connection Parameter

When defining a new block, a tolerance is used to identify the connection of this block between other blocks. This tolerance is automatically defined according to the geometry characteristics of the current block, and the users can scale this tolerance by setting the block connection parameter. The larger this parameter is, the easier the connection between two blocks will be built, and vice versa.

To set this parameter, first select a block, and then select “**Set Connection Parameter**” from **Block** panel or “**Settings...**” from **Generation** toolbar.

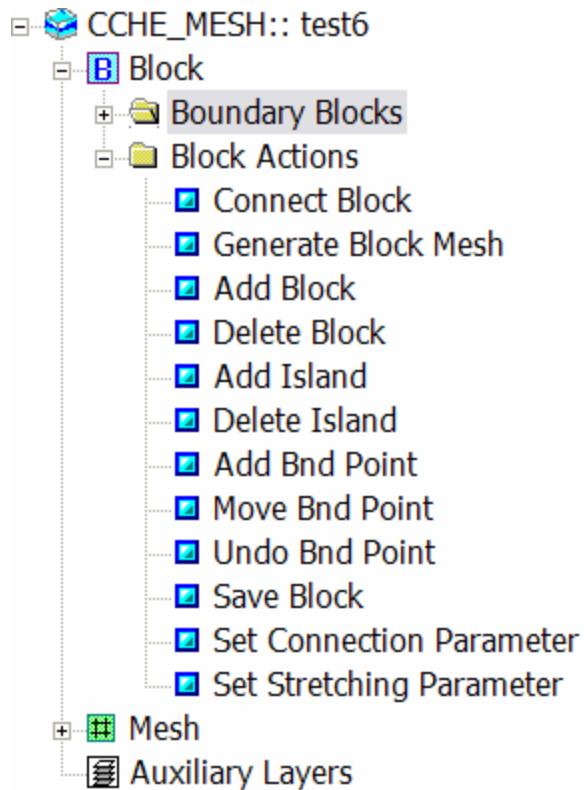


Figure 4-33

4.3.8 Edit Block Boundary

In the CCHE-MESH, you can also use the **Block Editing** tools to edit the existing block boundaries. You can **delete block**, **delete island** and **move the boundary control points**.



You can edit either the blocks from an existing block boundary file (*.mesh_mb) or the current existing blocks. In the first case, you need to load a block boundary file by selecting **Open Boundary File...** from **File** menu.

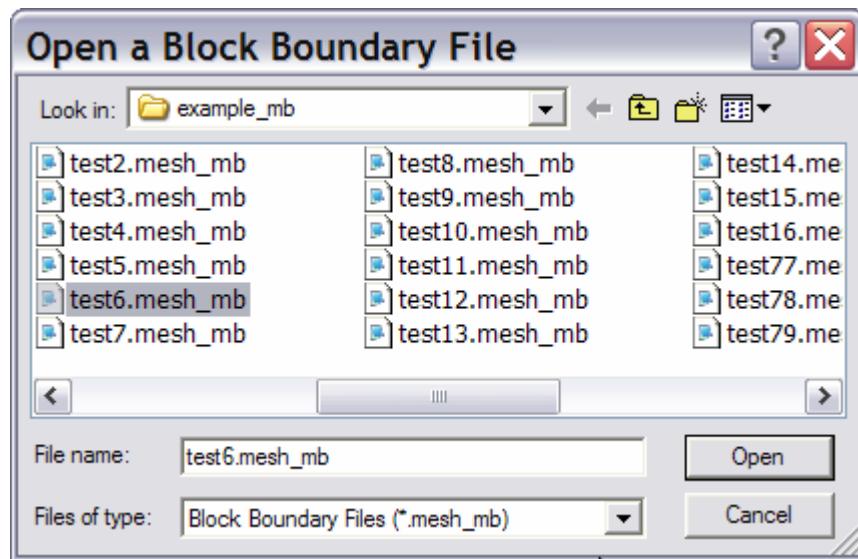


Figure 4-34

- To **delete a block**: click first, then click anywhere within the block you want to delete. If the block you want to delete has more than one connected blocks, you need to delete its neighbors before deleting this block. For example, as shown in Figure 4-28, if you want to delete block 13, you need to delete block 14 and 15 first.

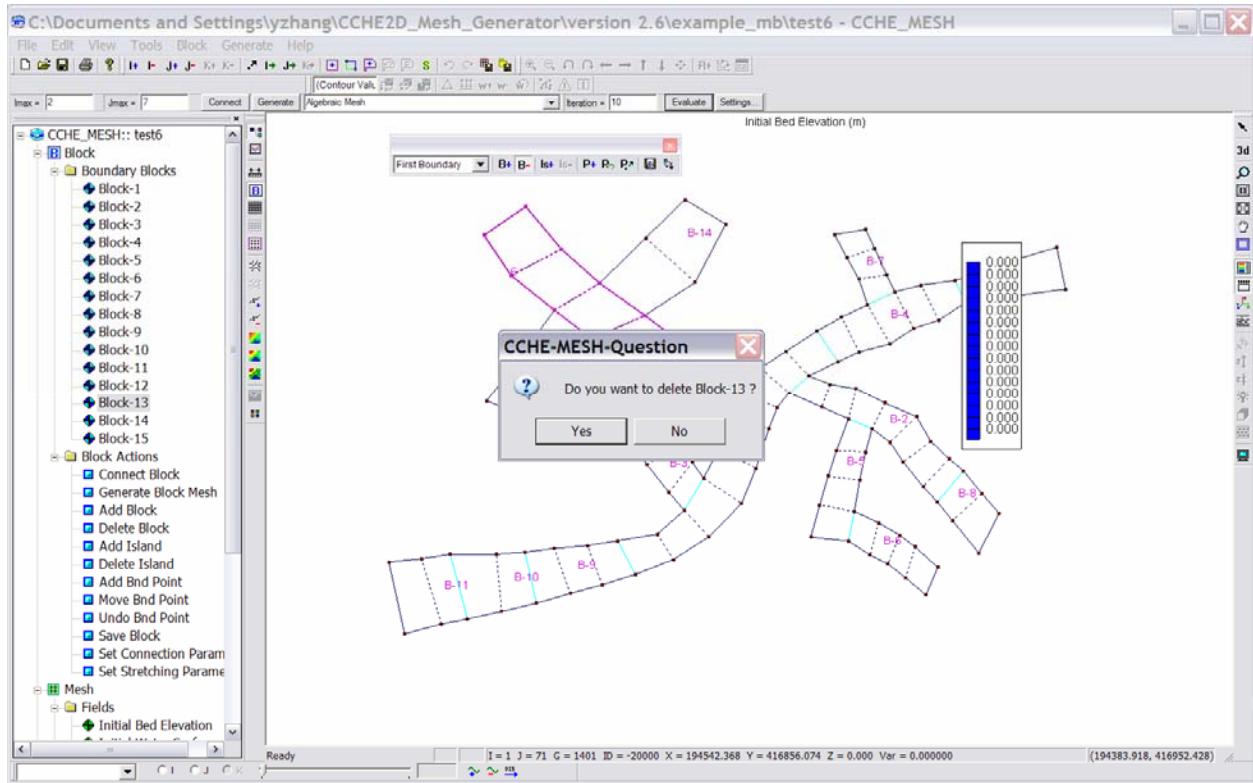


Figure 4-35

- To **delete an island**: select first, and then click any place within the island you want to delete.

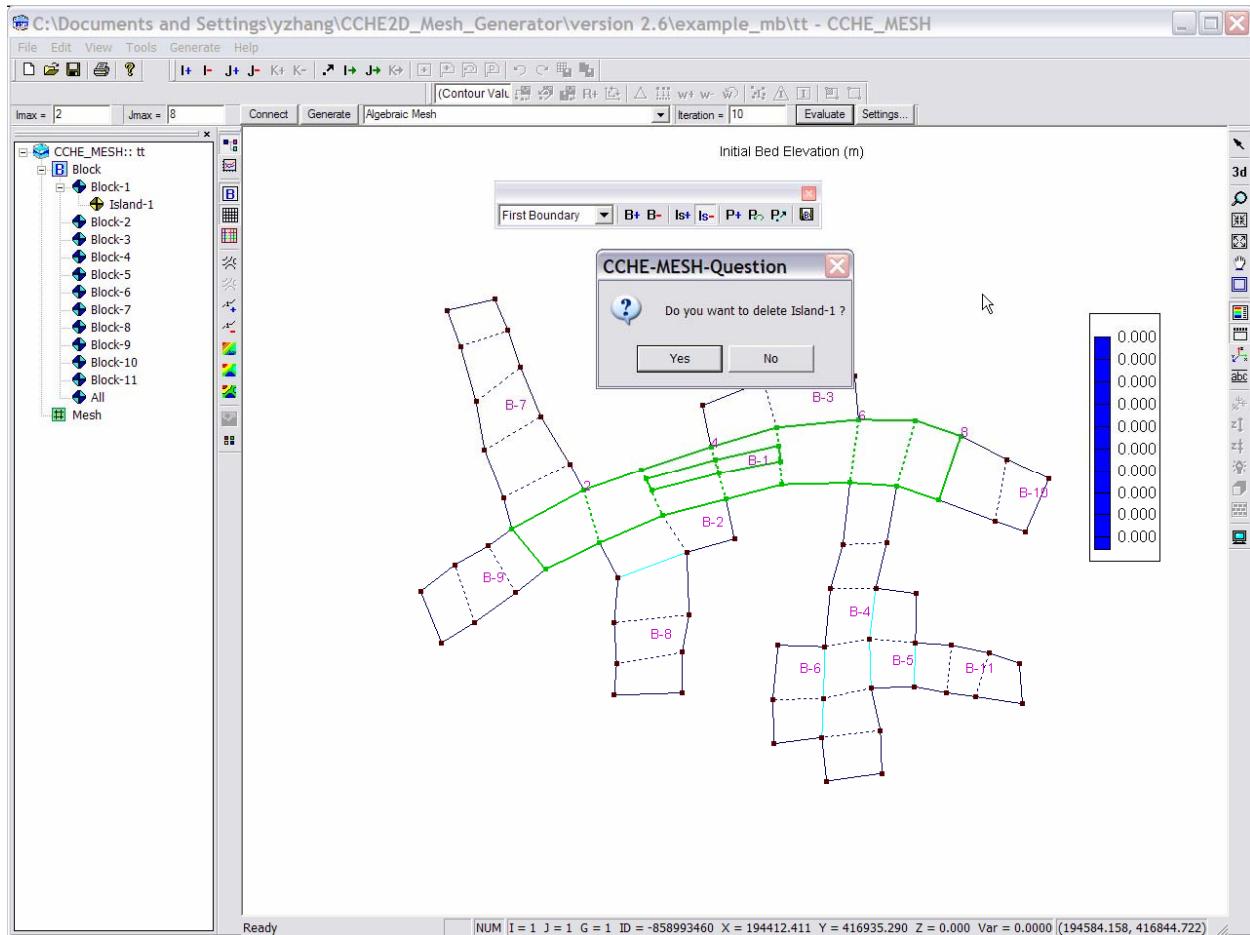


Figure 4-36

- To move boundary control points:
 - Select a block from **Block** group in **Project view** or by clicking anywhere within the block.

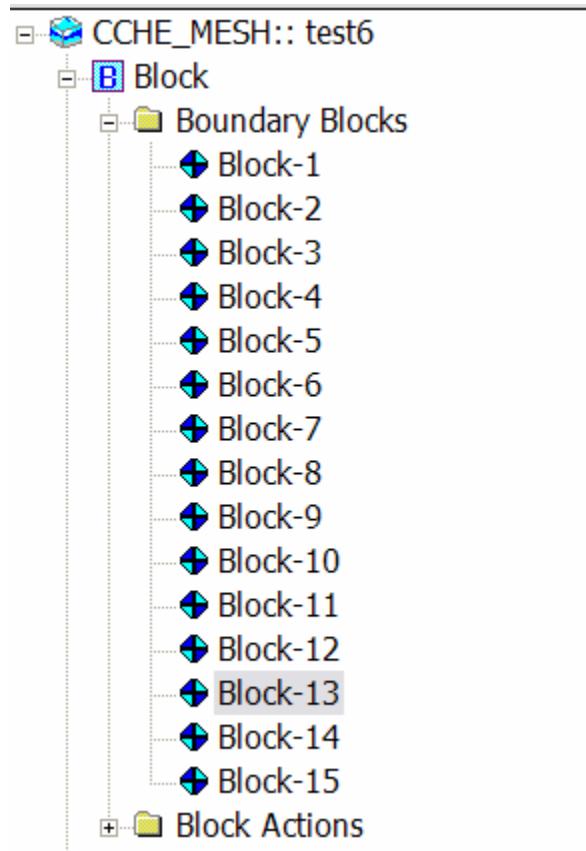


Figure 4-37

- Select on **Block Editing** Toolbar.
- Click the control point in the current selected block you want to move and hold it, then drag it to the desired place and release it.
- **Note:** there is **NO** undo function associated with this editing option, so be careful when moving the points. A safe way is to save the changes to another file without overwriting the original file.

4.3.9 Limitations of Multi-block Definitions

The multi-block scheme used in CCHE-MESH can greatly alleviate the difficulties of mesh generation in complex geometries, but they still have limitations or rules.

In one block, the basic rules are listed below and Figure 4-38 illustrates these rules:

- Equal number of control points along two boundaries (first and second) for both block and island
- The control points of island must be one-to-one corresponding to those of block
- Same orientation for block and islands
- Multiple islands can be created in the transversal direction in one block

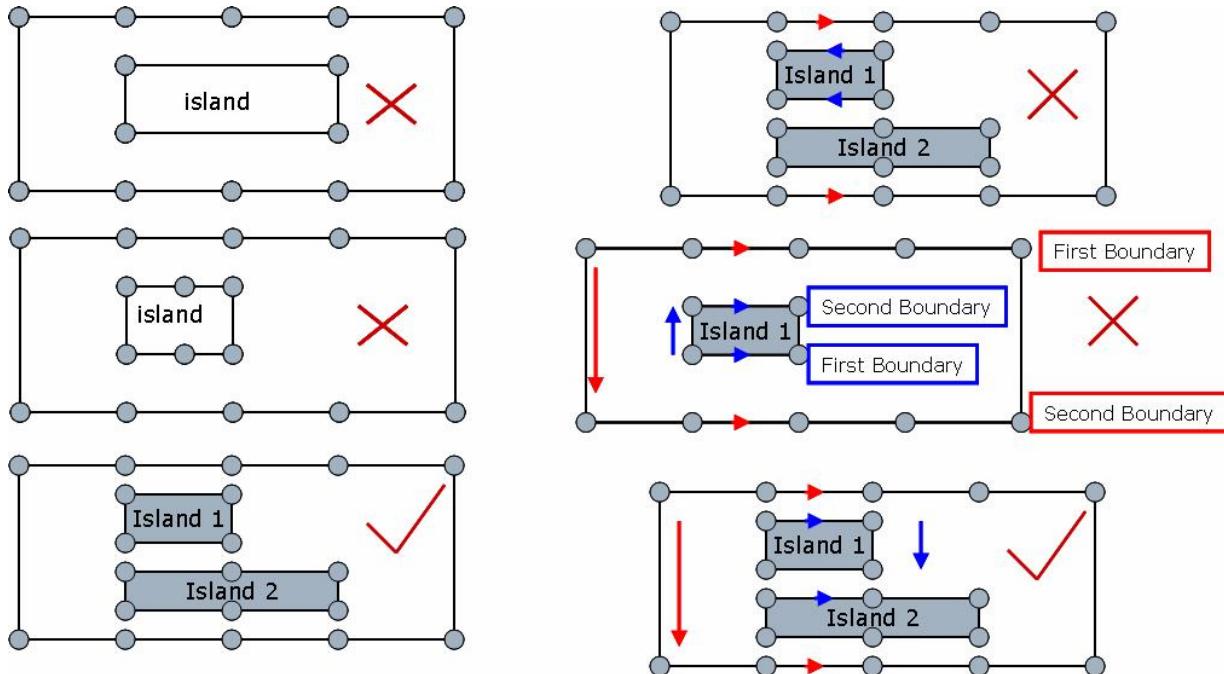


Figure 4-38 Basic rules in one block

For multiple blocks, the following basic rules are illustrated by Figure 4-39.

- An interface (blue line) contributed by only two blocks
- An interface consists of at least two points
- No interface points shared by more than 3 blocks
- No limitations on the number of blocks and islands

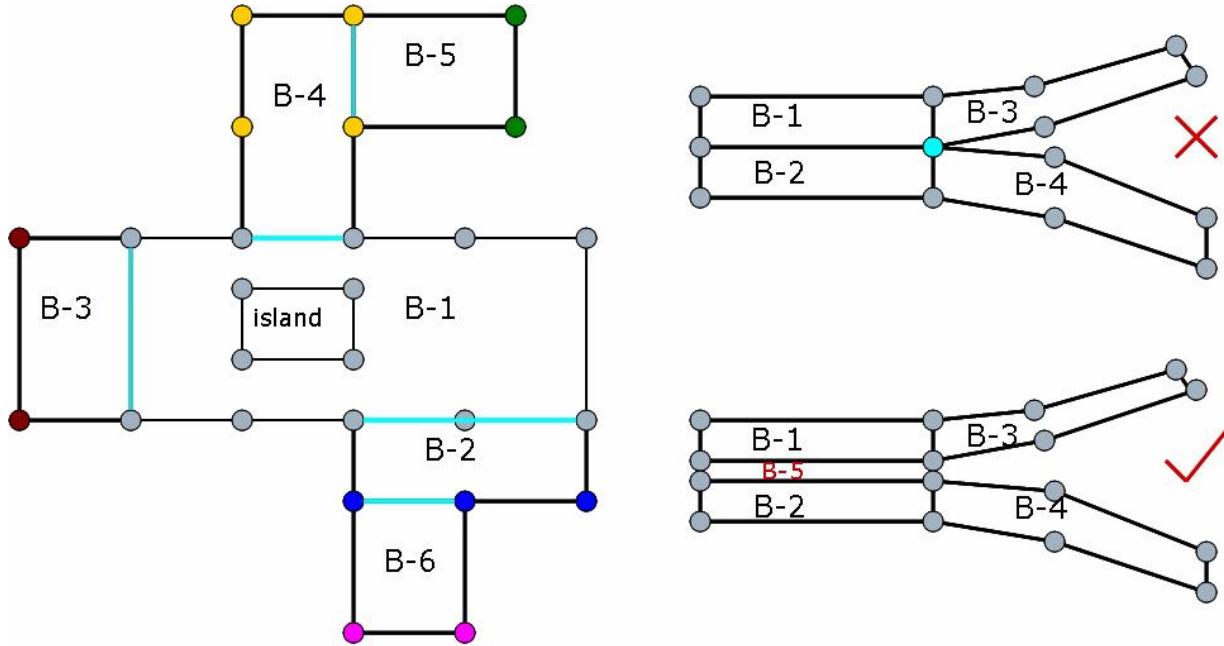


Figure 4-39 Basic rules of multi-blocks

These rules must be followed when defining blocks, otherwise the generator may fail.

4.4 Generate Mesh

One algebraic method and seven numerical methods have been integrated into the CCHE-MESH to generate quality meshes.

4.4.1 Generate Algebraic Mesh

To generate an algebraic mesh, a mesh block boundary file is needed. As described in detail in section 4.3, this file can be created from a topography map or a topography database.

Basically there are three steps to generate an algebraic mesh.

1. Open a mesh block boundary file or define block boundaries.

- Select **Open Boundary File...** from **File** menu to load a block boundary file.

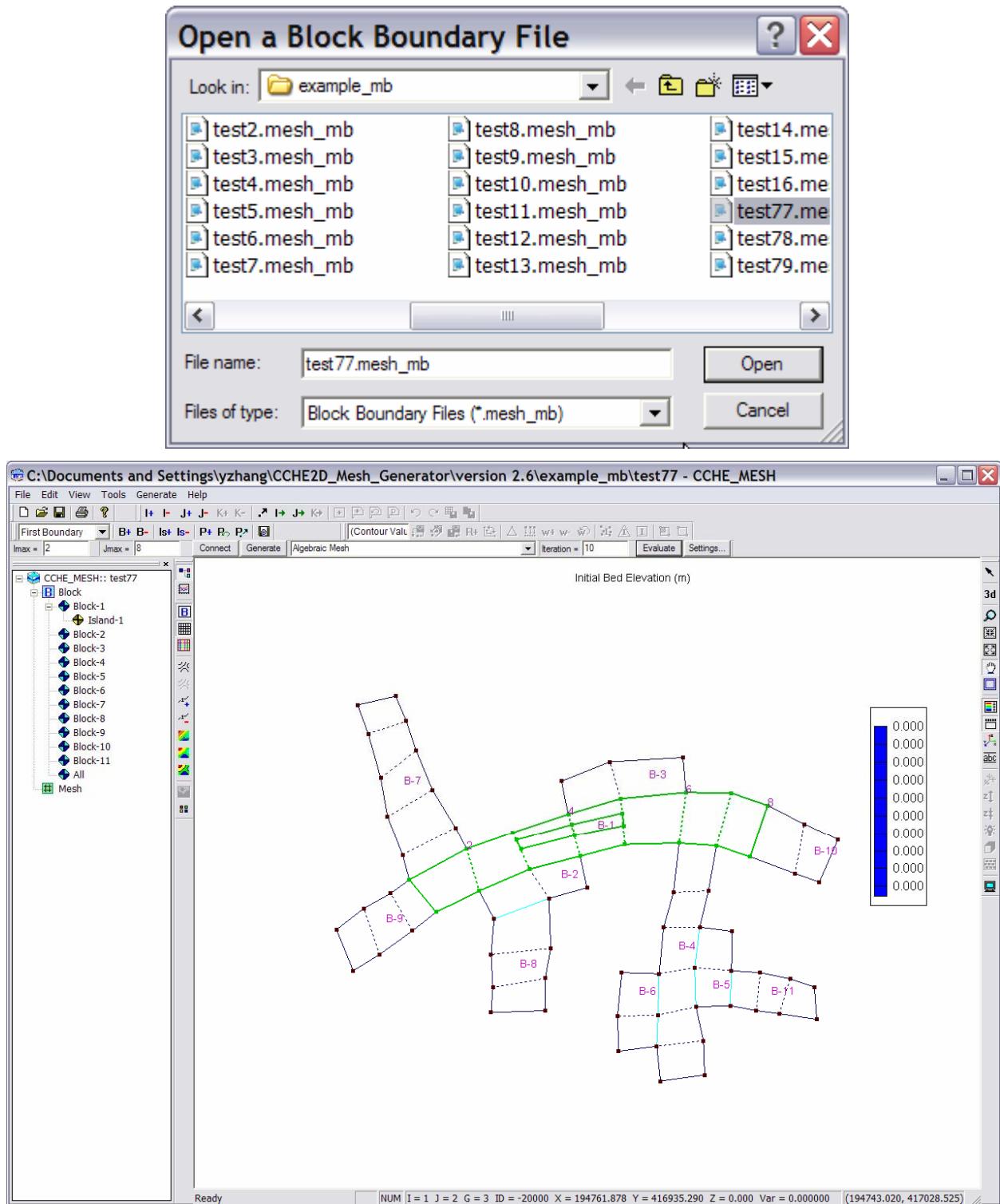


Figure 4-40

2. Specify mesh information.

- The mesh information includes, current block number, mesh size (Jmax number and Imax number).
- Select current block from **Project** view or by clicking any place within the desired block. **NOTE: the block-1 must be the first block for mesh generation.**

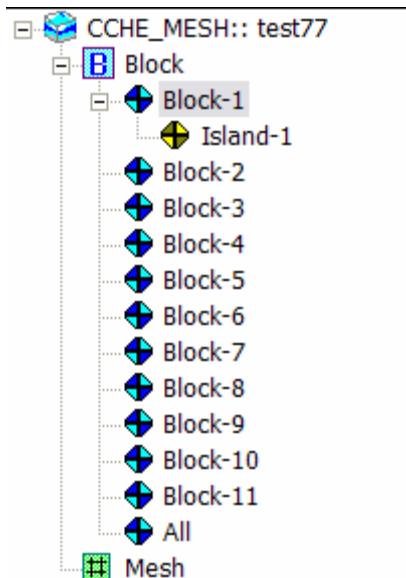


Figure 4-41

- Specify Jmax number and Imax number. 

- Select **Algebraic Mesh** from method selector.



- Select **Generate on Mesh Generation** toolbar or **Algebraic Mesh** in **Generate** menu

3. Set Stretching Function Parameters for algebraic mesh generation.

- NOTE:** Before setting parameters for one block, you need to generate an algebraic mesh first for this block.

- To set parameters, click **Settings...** on **Mesh Generation** toolbar to activate **Stretching Function** for algebraic mesh generation.

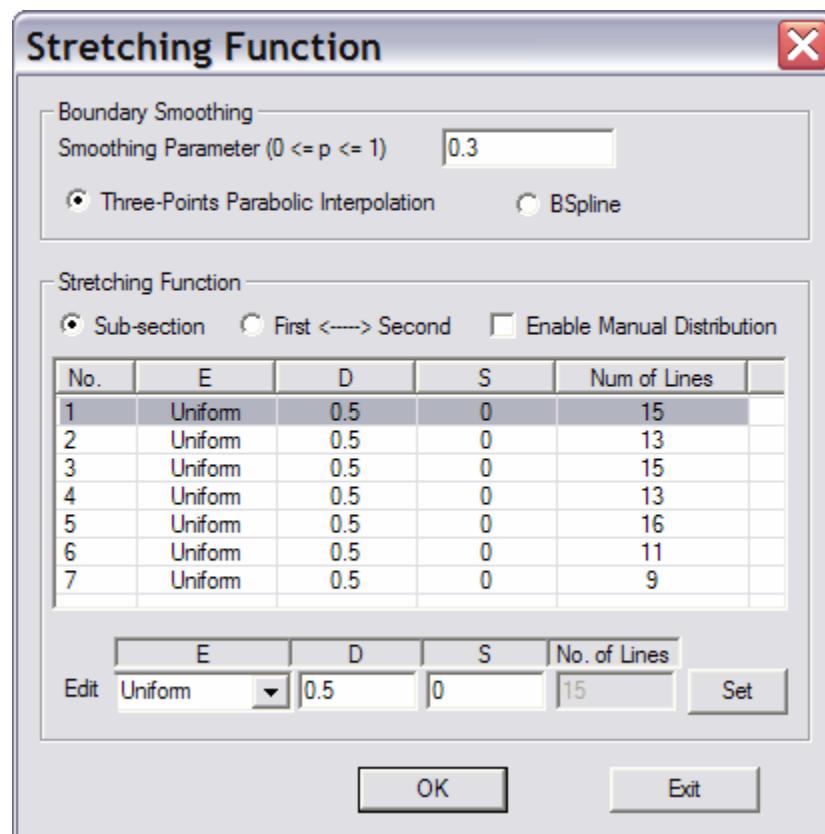


Figure 4-42

- In **Stretching Function** window, the parameters are divided into two groups: parameters for **Boundary Smoothing** and parameters for **Stretching Function**. In **Boundary Smoothing**,
 - There are two options to smooth boundary: **Three-Points Parabolic Interpolation** and **B-Spline**.
 - The **Smoothing Parameter** is used to control the smoothing effects. If $p = 0$, there is no smoothing at all; and, if $p = 1$, the maximum smoothing effects will be applied.
 - Three-Point Parabolic Interpolation:** Select this option if you want to use the three-point parabolic function to interpolate the boundary nodes. The

advantage of this option is that it can keep the original shape of the domain, and the drawback is that sometimes the boundaries are not sufficient smooth.

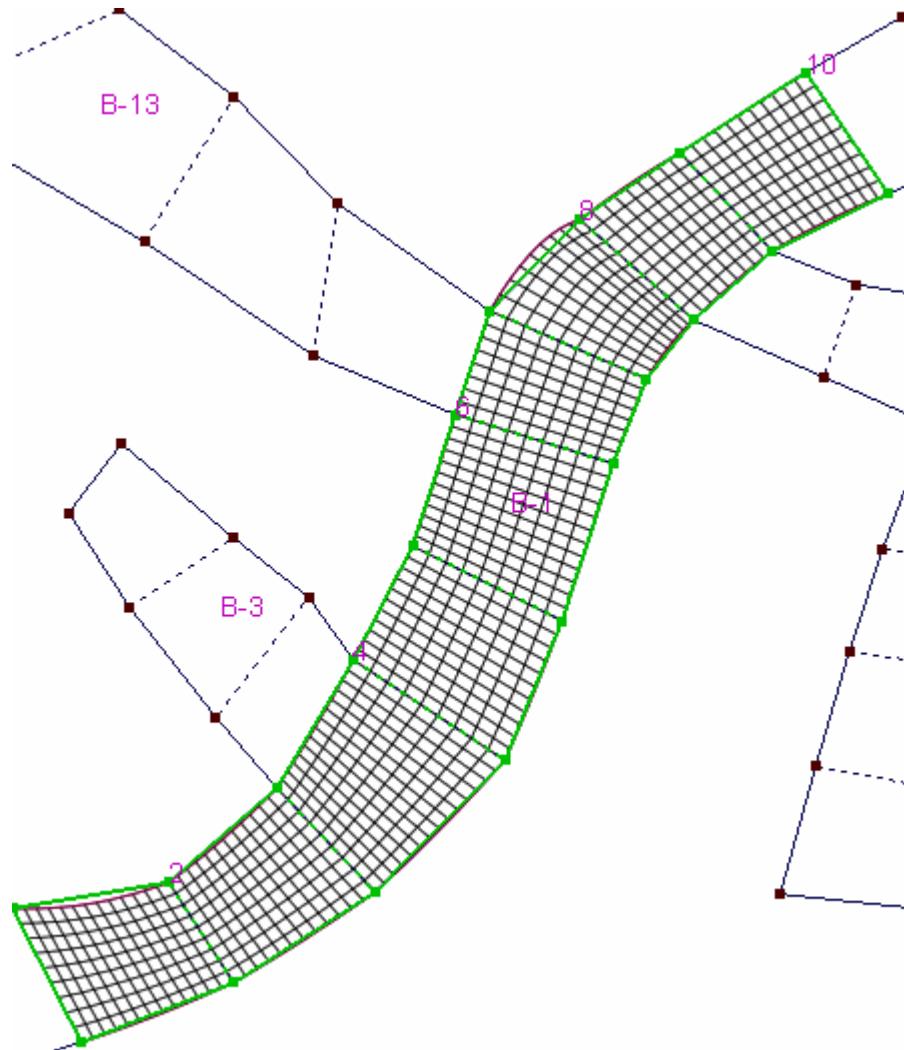


Figure 4-43

- **Using B-Spline:** Select this option if you want to smooth the boundary with B-Spline. This option is very stable. However, the smooth curve will not pass through all control points, so for some cases it cannot keep the original shape of the domain.

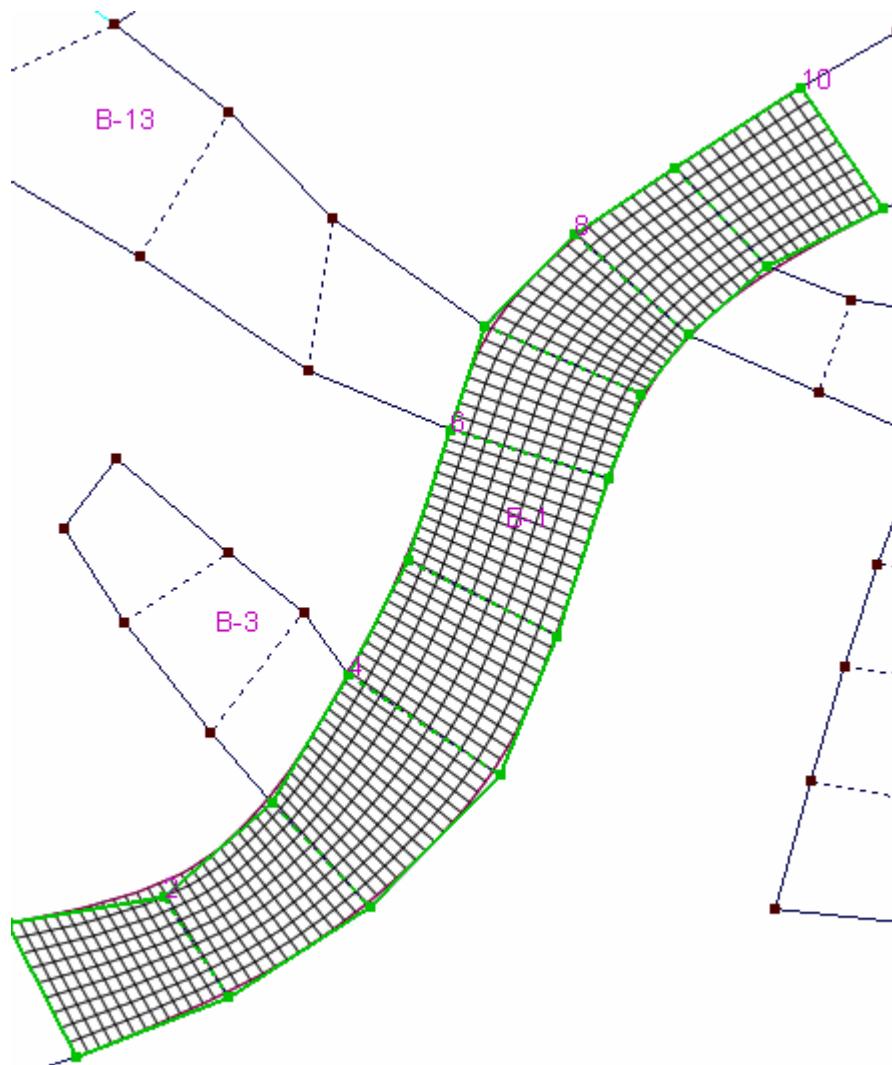


Figure 4-44

- For **Stretching Function**,

- The CCHE-MESH uses a two-boundary method to generate algebraic mesh. As shown in Figure 4-45, the shape of a block domain is controlled by the “top” (**First**) boundary and “bottom” (**Second**) boundary. The whole domain is divided into sub-sections by the control points.
- The stretching function can be applied for **Sub-section** and **First \leftrightarrow Second**.
- The main parameters are **E**, **D**, and **S**. Please refer to section 2.2.2 for the details of the **EDS stretching function**. **E** determines the characteristic of

the stretching, **Uniform (E=0)**, **Contraction (E=-1)** or **Repulse (E=1)**; **D** ($0 \leq D \leq 1$) determines the relative location where the stretching happens; **S** ($0 \leq S \leq 10$) determines the stretching degree.

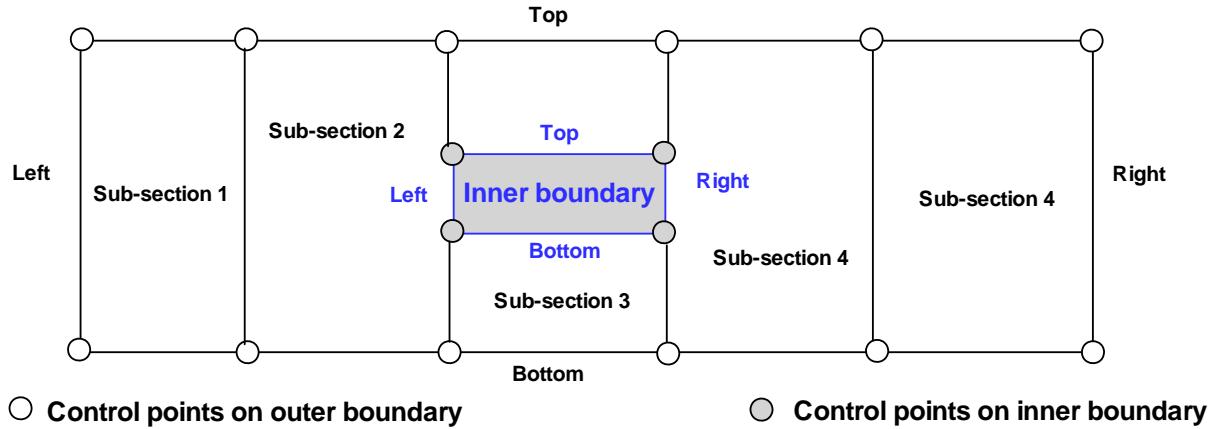


Figure 4-45

- **Sub-section:** You can specify **E**, **D**, and **S** parameters for each sub-section. If you want to specify the **Number of Lines** in each sub-section, please check **Enable Manual Distribution**.
- **First $\leftarrow\rightarrow$ Second:** You can use the stretching function to control the distribution of mesh nodes in the direction from the first boundary to the second boundary (top to bottom). Only **E**, **D**, and **S** parameters can be set for this option.

4. Generate mesh for the current block.

- If you want to see the immediate effects of the parameters you set, click **OK**. A mesh will be generated based on the current settings. The **Stretching Function** window will not be closed.
- If you want to exit the window, please select **Exit**.

5. Generate meshes for other blocks.

- Repeat from step 2 to step 4 to generate meshes for other blocks.
- **Note that** because of the characteristics of the structured mesh, the maximum I lines or the maximum J lines or both of the block will be determined by the neighboring blocks in order to make the interfaces between neighbors compatible with each other.

For example, in Figure 4-46, the maximum of I lines of block-4 will be the same as that of block-1.

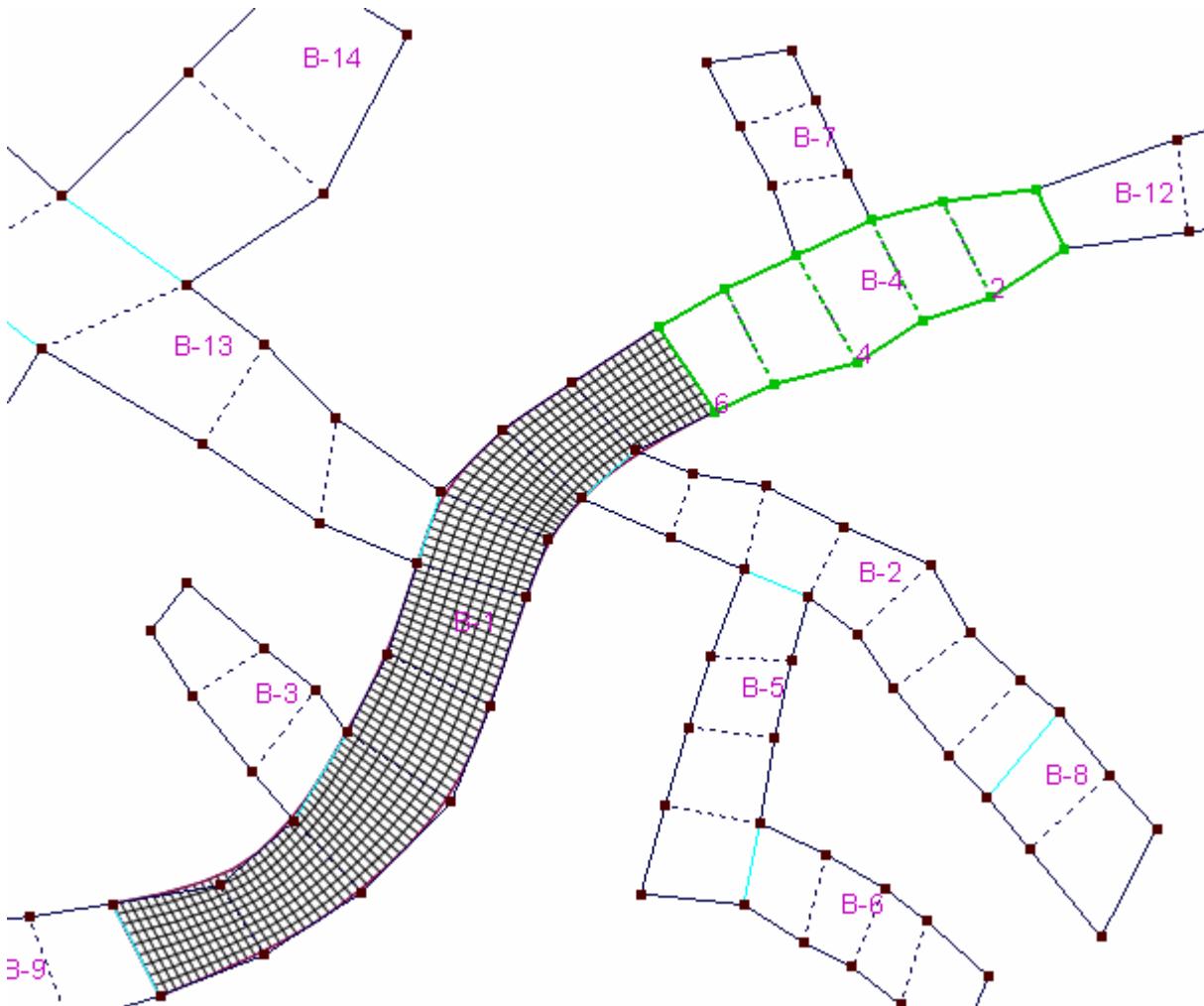


Figure 4-46

- For the multi-block domain as shown in Figure 4-42, you can generate meshes block by block. As long as the boundaries of the generated blocks are neighbored each other, you can connect them into one whole mesh by selecting **Connect** on **Mesh Generation** toolbar or **Connect Block** from **Project view**. For those blocks without connections, you cannot connect them.

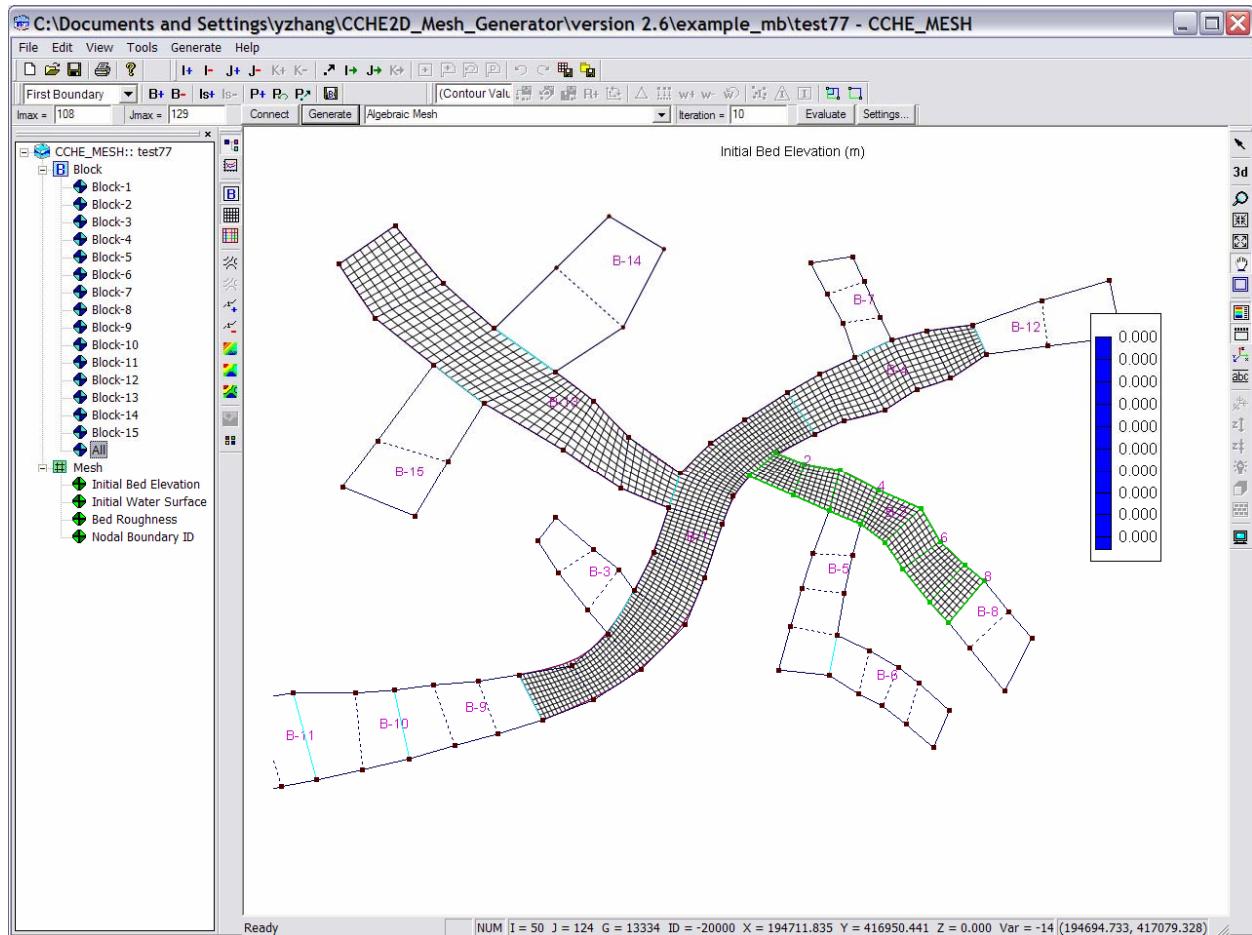


Figure 4-47

6. **Save mesh:** After all meshes are generated, select **Save as Geometry File...** from **File** menu or on **Mesh Editing** toolbar.

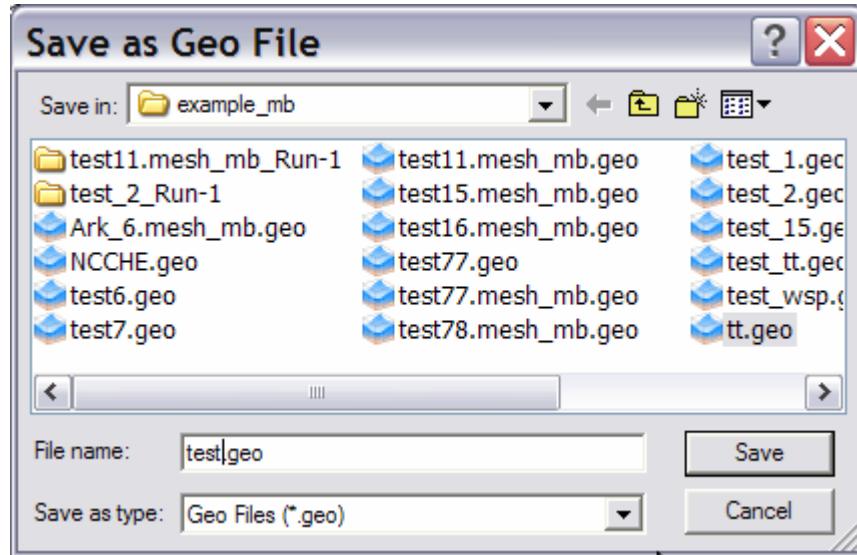
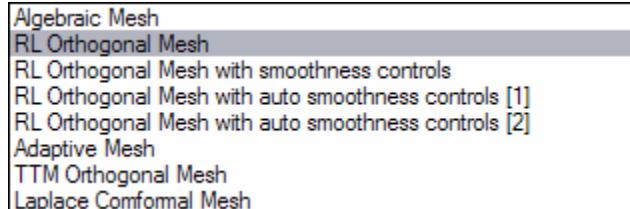


Figure 4-48

4.4.2 Generate Quality Mesh

After an algebraic mesh is generated, you can further improve the mesh quality using the numerical methods. The CCHE-MESH provides seven numerical methods to generate quality mesh. Please refer to section 2.3 for technical details of these methods.



To generate quality meshes, there are six steps to follow.

1. Specify the Iteration Number.

- Specify the iteration number in
- Tip:** You don't have to set a big iteration number for one method. Instead you'd better set a small number and generate several more times.

2. Specify the Effect Area.

- The default effect area is the whole domain. That is, I lines are from 1 to Imax and J lines are from 1 to Jmax.

- You can also apply the generation method to a specific area of the domain. To fulfill it, select  first, and then click two different points on the mesh to define a rectangular area.

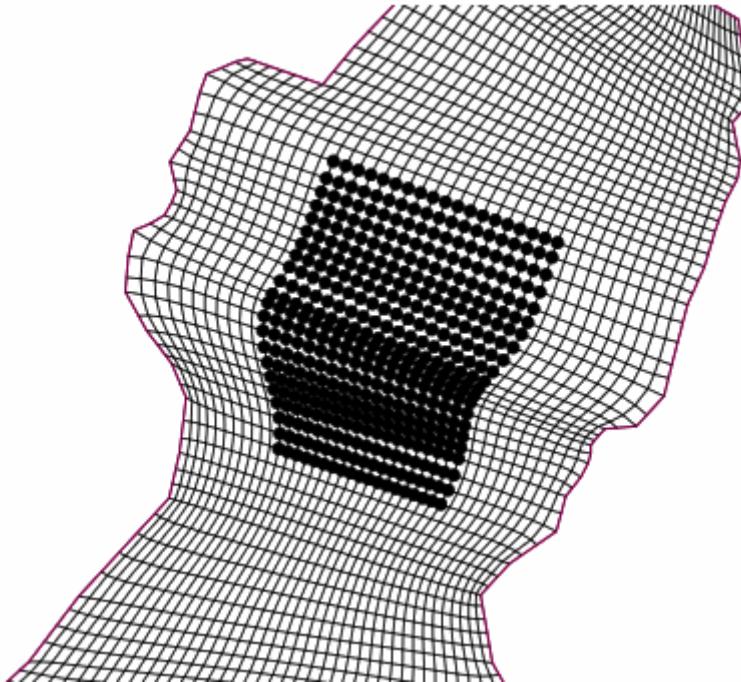
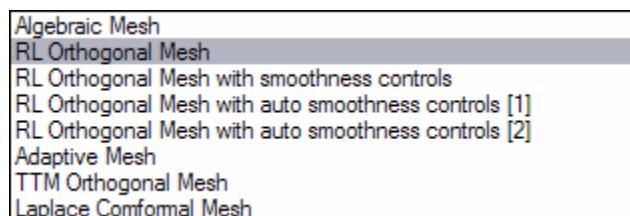


Figure 4-49

- To go back to the default effect area, select  or **Reset Effect Area** from **Tools** menu.

3. Select Generation Method.

- Select a generation method from method selector.



4. Specify the Generation Parameters if applicable.

- For method **RL Orthogonal Mesh with smoothness controls**, you can set the smoothing parameter. The larger it is, the smoother the mesh will be; and vice versa. This parameter is no use for other numerical methods.

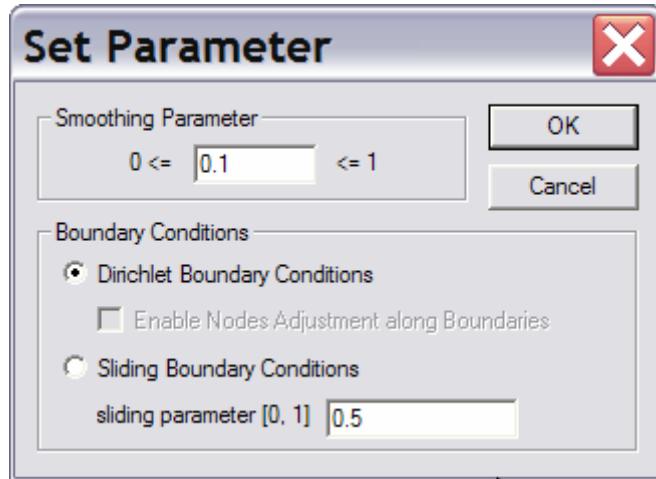
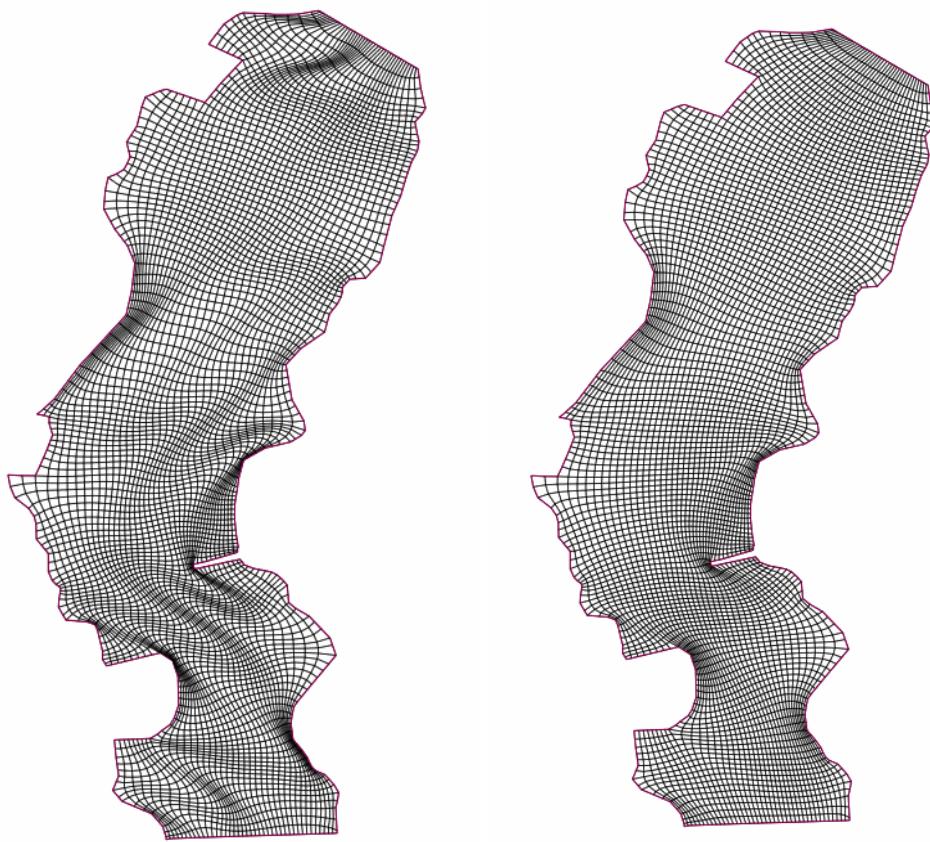


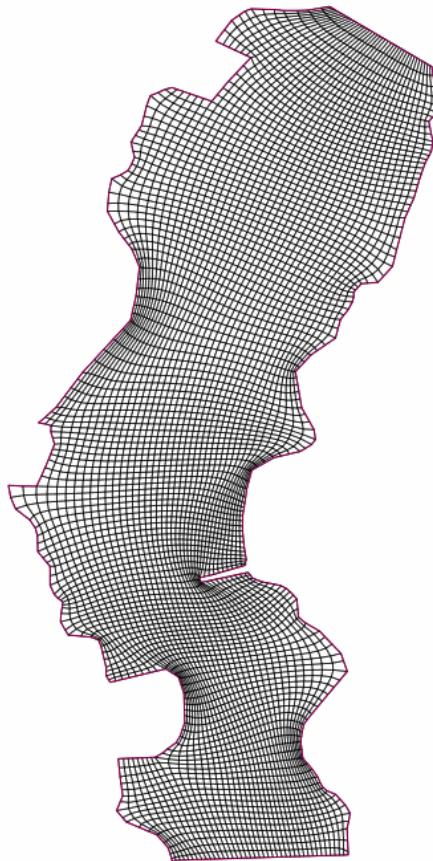
Figure 4-50

- Two boundary conditions are available: **Dirichlet Boundary Conditions** and **Sliding Boundary Conditions**. By default, the Dirichlet boundary conditions will be used for the whole boundaries. If the users select the sliding boundary conditions, they will be applied to the boundaries according to the boundary curvatures. The boundary with high or complex curvatures will be applied with highest weighting, and the boundary with simple curvature will have low weighting.
- **Recommendation:** For natural rivers with irregular boundaries, test cases have shown that the method of **RL Orthogonal Mesh with smoothness controls** is very reliable and robust to generate quality meshes.

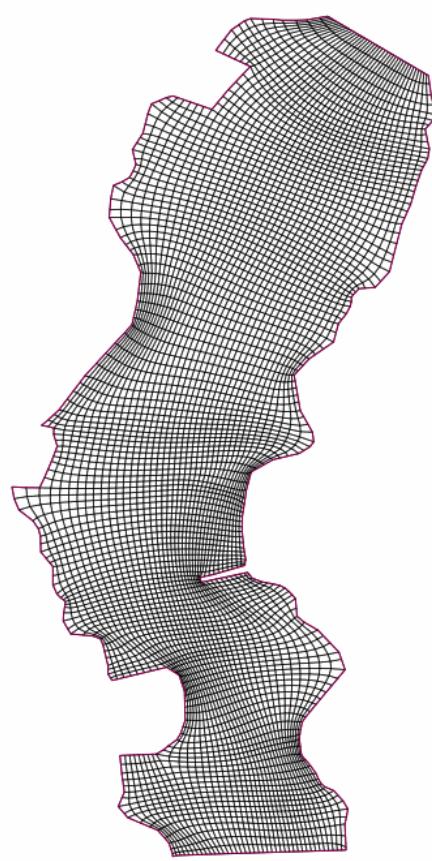


(A) RL orthogonal mesh

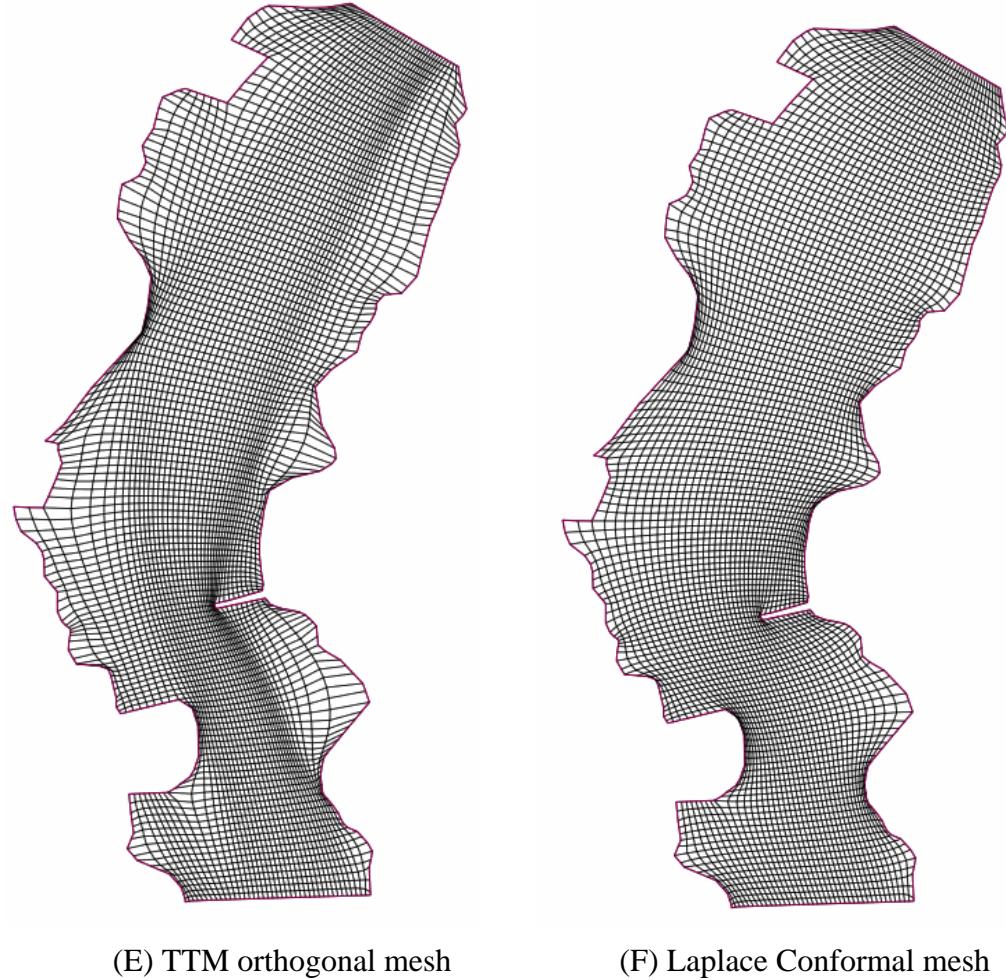
(B) RL orthogonal mesh with
smoothness control



(C) RL orthogonal mesh with
Auto smoothness control [1]



(D) RL orthogonal mesh with
auto smoothness control [2]



(E) TTM orthogonal mesh

(F) Laplace Conformal mesh

Figure 4-51

5. Generate mesh.

- Select **Generate** on **Mesh Generation** toolbar, and the mesh based on the current settings will be generated. Figure 4-51 compares different mesh generation methods in a natural river channel.
- You can always undo the change you made by selecting on **Mesh Editing** Toolbar.
- You can also restore the changes you made by selecting on **Mesh Editing** Toolbar.

- If you are satisfied with the generated mesh, you can select **Save as Geometry File...** from **File** menu or  on **Mesh Editing** toolbar.

4.4.3 Generate Adaptive Mesh

The CCHE-MESH is capable of generating adaptive meshes. As shown in Figure 4-50, a curved natural river is used to illustrate the adaptive mesh generation.

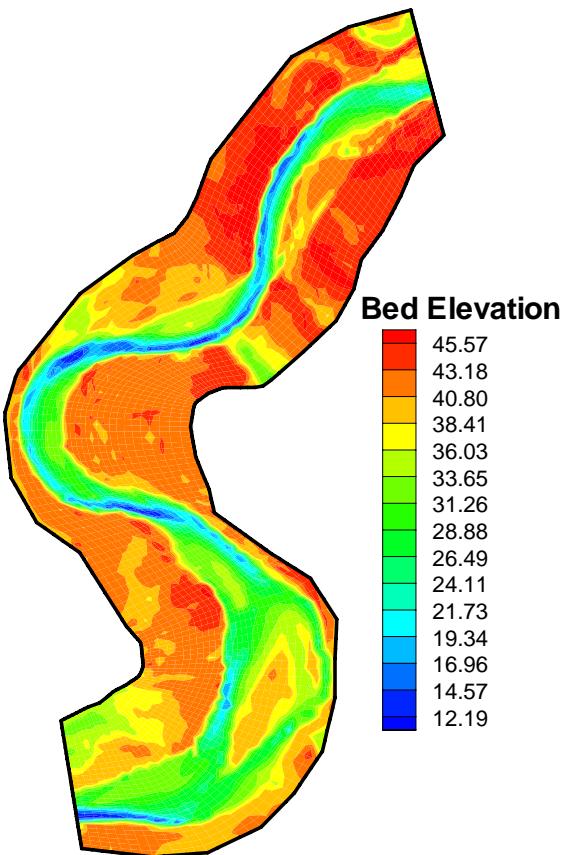


Figure 4-52 A Curved Natural River

For **Adaptive Mesh** generation, four steps are needed.

1. Generate an algebraic mesh or load an existing mesh.
2. Select **the adaptive variable** from **Project** view.

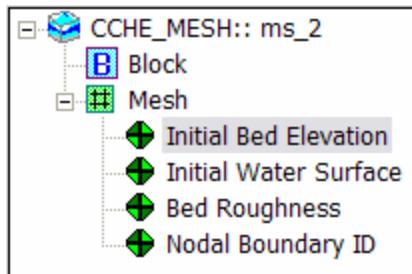


Figure 4-53

3. Set Iteration number.
4. Select Adaptive Mesh from method selector

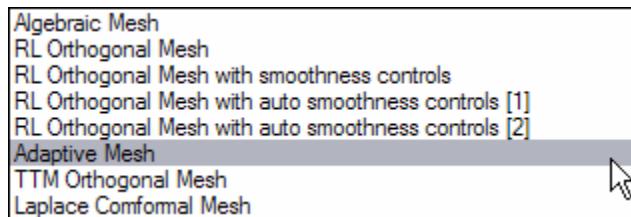


Figure 4-54

5. Select Generate on Mesh Generation toolbar.

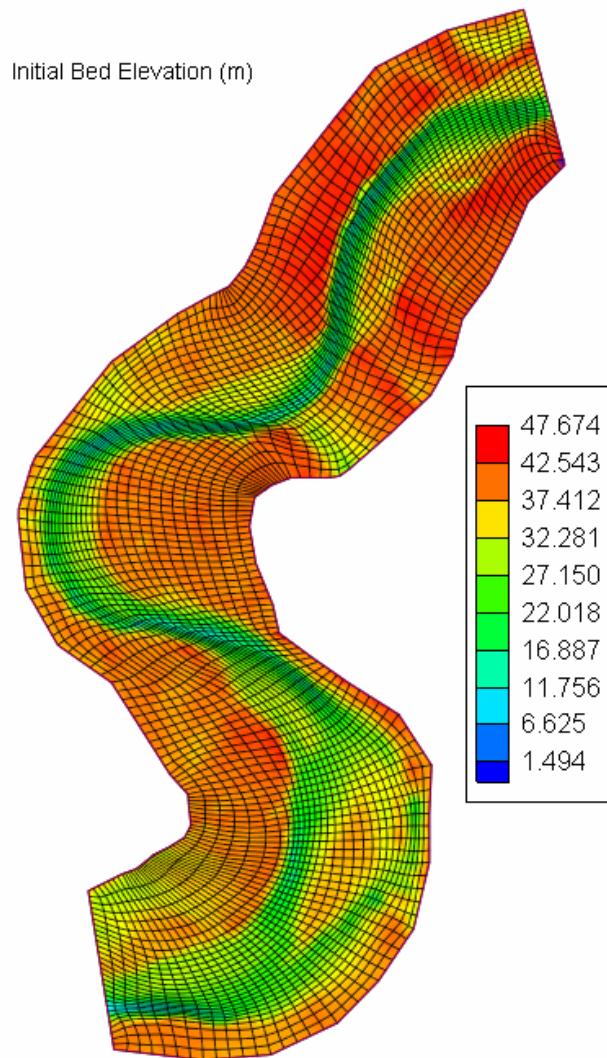


Figure 4-55

4.5 Evaluate Mesh

Once a mesh is generated, you can get an evaluation report of the mesh. In the CCHE-MESH, the Maximum Deviation Orthogonality (MDO), Averaged Deviation from Orthogonality (ADO), Maximum grid Aspect Ratio (MAR), and Averaged grid Aspect Ratio (AAR) are used to evaluate a mesh. For details please refer to section 2.4.

You can select **Evaluate on Mesh Generation** toolbar to get the evaluation report.

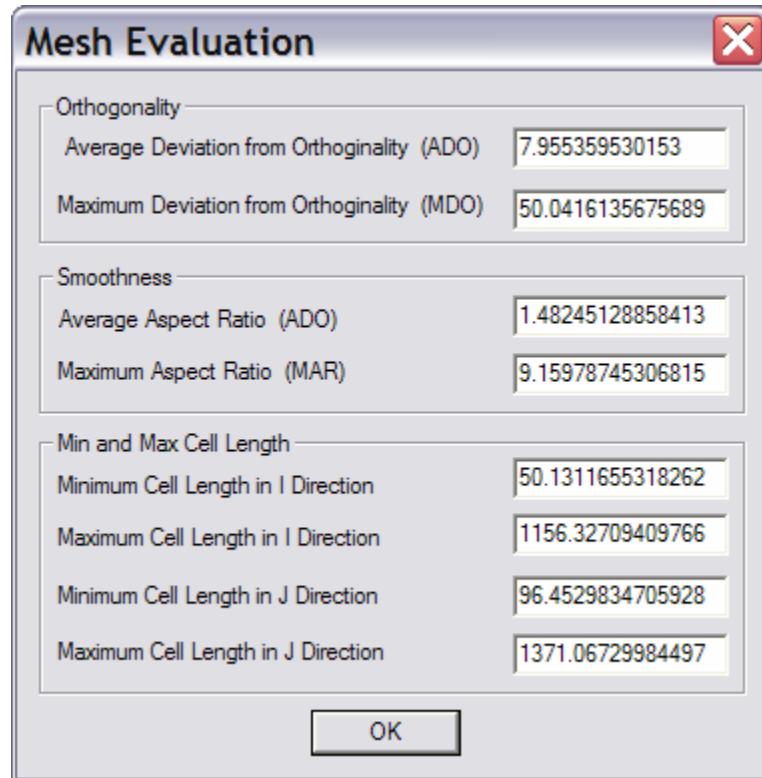


Figure 4-56 Evaluation Report

In **Mesh Evaluation**, three groups information are provided, namely, **Orthogonality**, **Smoothness** and **Min and Max Cell Length**. For orthogonality, the smaller the ADO and MDO are, the better the orthogonality of the mesh is, while for smoothness, the closer to 1 the AAR and MAR are, the better the smoothness of the mesh is. The min and max grid lengths provide you the information of the general resolution of the mesh.

4.6 Bed Elevation Interpolation

To make a mesh practically useful, the bed elevation needs to be interpolated from a topography database (*.mesh_xyz). The CCHE-MESH Mesh Generator provides three bed interpolation algorithms for different types of database, namely, **Random Interpolation**, **Triangulation Interpolation**, and **Structured Interpolation**. Please refer to Chapter 3 for details.

As described in Chapter 3, the random interpolation is used for the random database, while the structured interpolation is for the structured database (*.mesh_mcs). As for the triangulation interpolation, it can be used for both types of database. The CCHE-MESH also provides a way to refine the structured database for more accurate interpolation. You can also smooth the bed to remove the oscillation of the interpolation.

The above actions on bed can be selected from the **Tools** toolbar and menu.



4.6.1 Bed Interpolation

As shown in Figure 4-57, a symmetric database and an asymmetric database will be used to illustrate the effects of different bed interpolation methods.

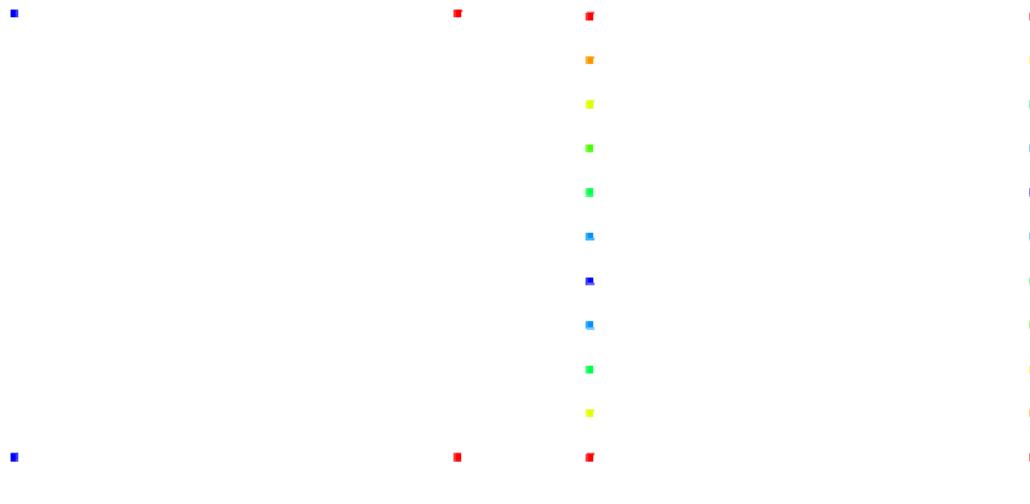


Figure 4-57

The measured survey cross sections (*.mesh_mcs) are also provided for the above two database in Figure 4-57. As can be seen, there are only two cross sections for both databases.

To interpolate the bed elevation, you need to follow the steps below.

- **Generate a mesh or load an existing mesh.**

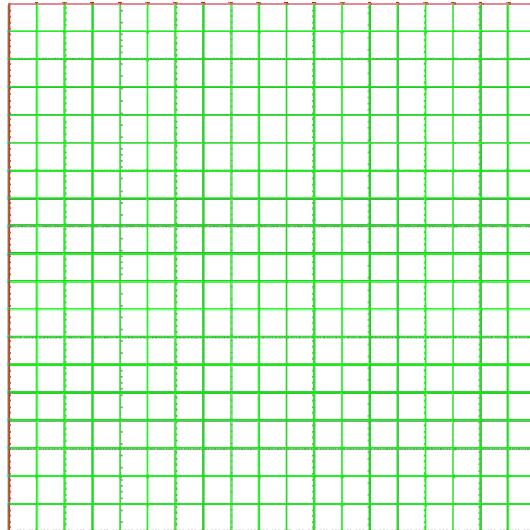
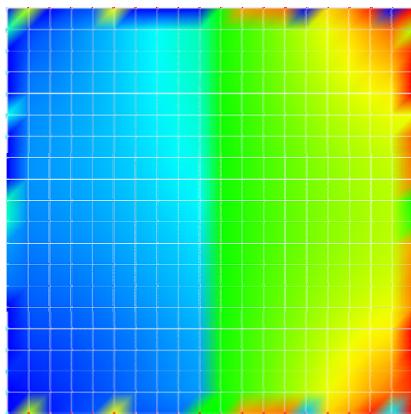


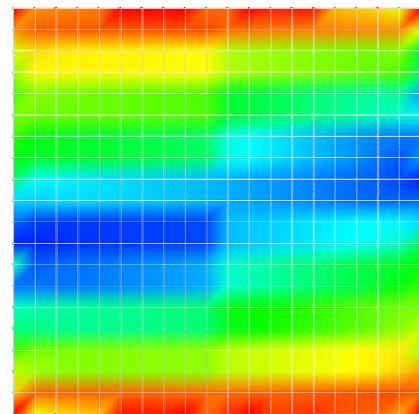
Figure 4-58

- Select an interpolation method.

- **Random Interpolation** : This method is used for the random database (*.mesh_xyz). It is fast but not stable when the database is sparse.



(A) Symmetric database

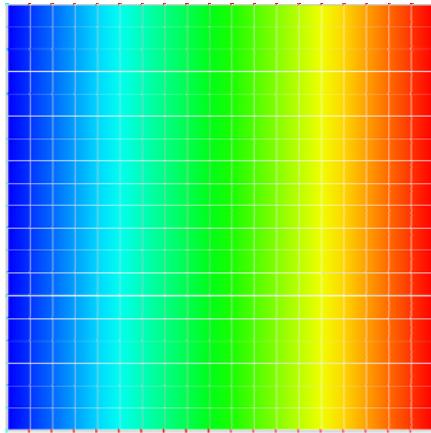


(B) Asymmetric database

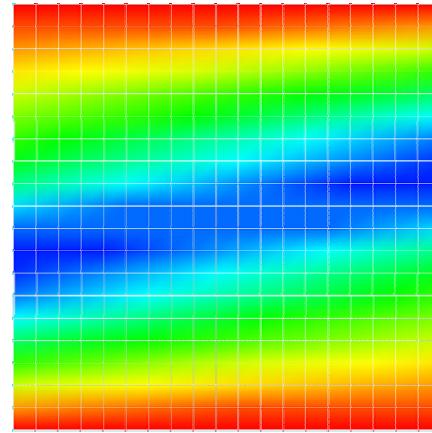
Figure 4-59 Random Interpolation

- **Triangulation Interpolation** : This method can be applied to both random and structured database. This method is very stable but it is slow and requires much more

computational efforts. It requires that the mesh nodes be within the database. It will not work on those outside of database.



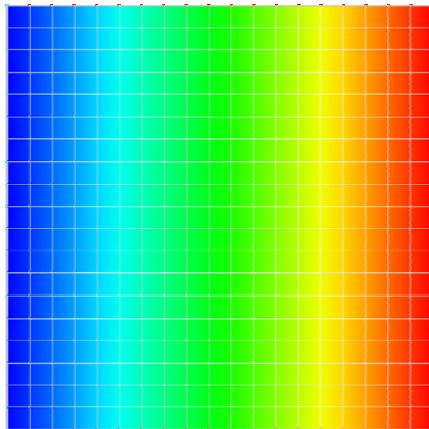
(A) Symmetric database



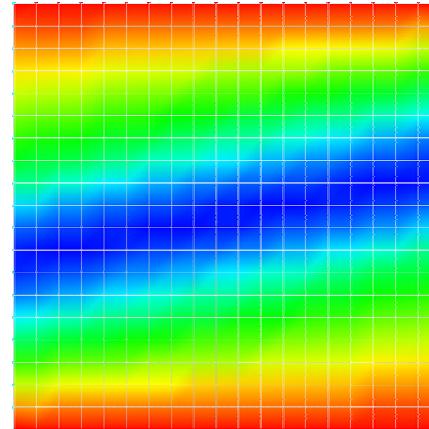
(B) Asymmetric database

Figure 4-60 Triangulation Interpolation

- **Structured Interpolation**  : This method needs the measured cross section file (*.mesh_mcs). It will refine the database first and then use planar interpolation method to interpolate the bed elevation from the refined database.



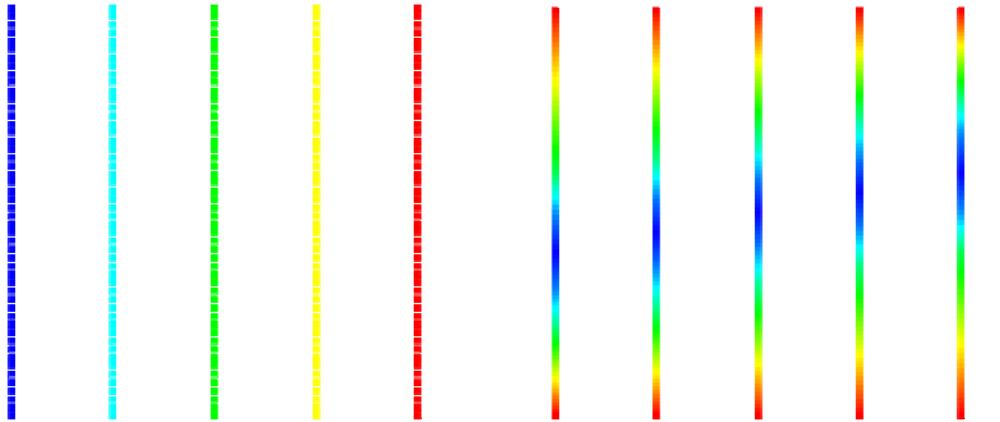
(A) Symmetric database



(B) Asymmetric database

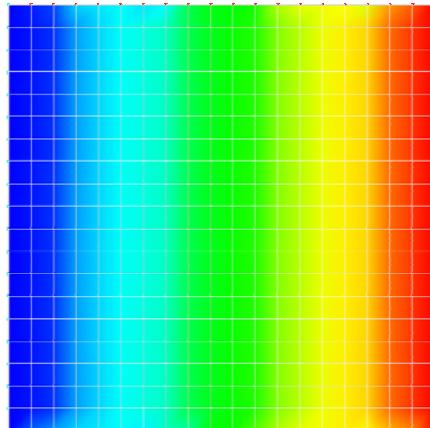
Figure 4-61 Structured Interpolation

- **Database Refinement:** If you have a measured cross section file (*.mesh_mcs), you can create a refined random database from it. As shown in Figure 4-56, with the refined database, the accuracy of interpolation can be improved.

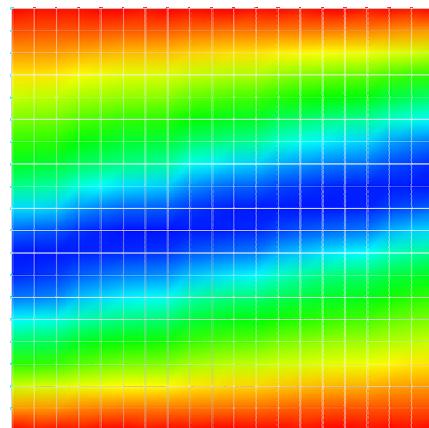


(A) Refined Symmetric database

(B) Refined Asymmetric database



(C) Using Refined Symmetric Database



(D) Using Refined Asymmetric Database

Figure 4-62 Random Interpolation

4.6.2 Bed Smoothing

For some cases, the numerical model may be sensitive to the bed topology. The non-physical bed oscillation introduced by the interpolation would affect the simulation results significantly. To reduce the oscillation, you can smooth the bed.

To smooth the bed,

- First **define the effect area**: the bed smoothing is usually applied to some local area because it's dangerous to smooth the whole domain which may change the original topology. To define a local area, select  first and then click two different points on the mesh to define a rectangular area.

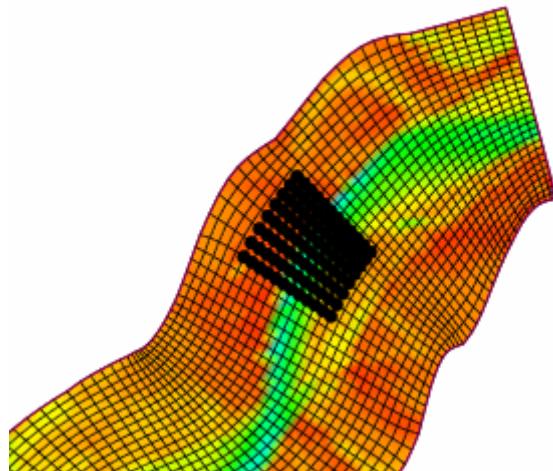
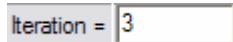


Figure 4-63

- **Set iteration number**  : you can set the iteration number to control the smoothing effect. The more iteration steps, the smoother the bed will be.
- **Smooth bed**: select **Initial Bed Elevation** from **Mesh** group in **Project view**, and then select **Smooth Bed** from **Bed Interpolation** submenu in **Tools** menu or  on **Tools** toolbar.

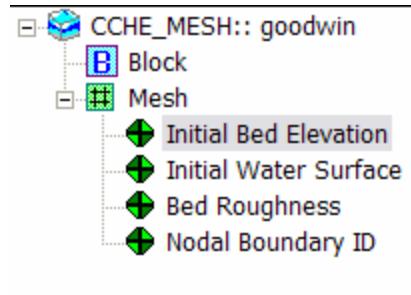
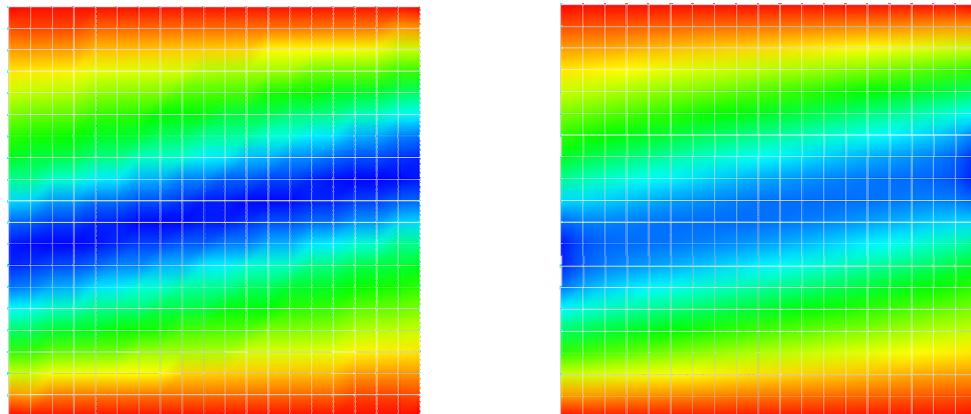


Figure 4-64

- Figure 4-65 shows the effects of the bed smoothing for the mesh using the random interpolation.



(A) Before Smoothing

(B) After Smoothing

Figure 4-65

4.7 Edit Mesh

With an existing mesh, you can further edit the mesh to satisfy your needs.

4.7.1 Edit Mesh

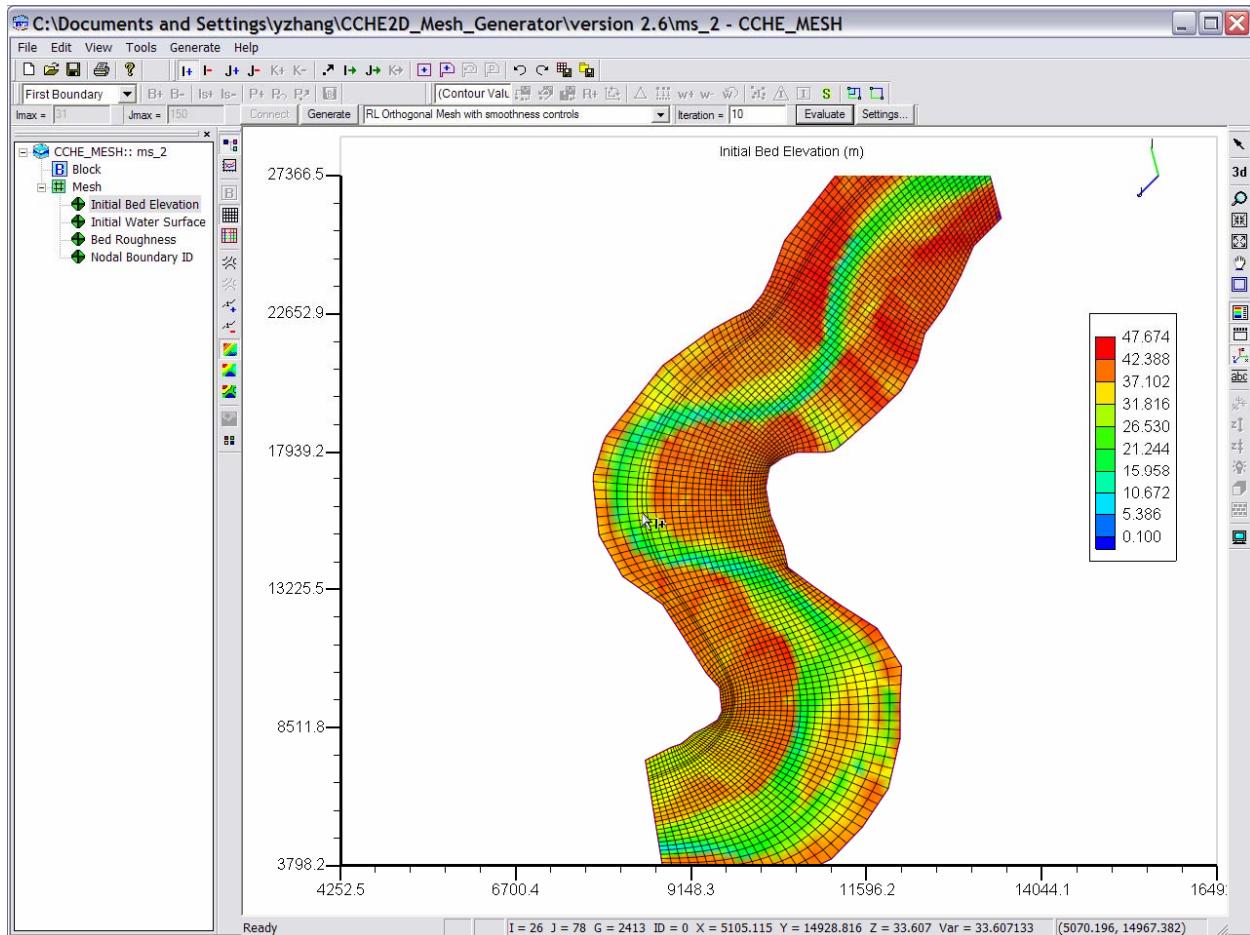
The CCHE-MESH provides five options for mesh editing: **Add Mesh Lines**, **Delete Mesh Lines**, **Move Mesh Lines**, **Move Mesh Node**, and **Change Field Values or Nodal Boundary ID**. To activate these editing options, you can access them either from **Mesh Editing** toolbar or from menu **Edit**.

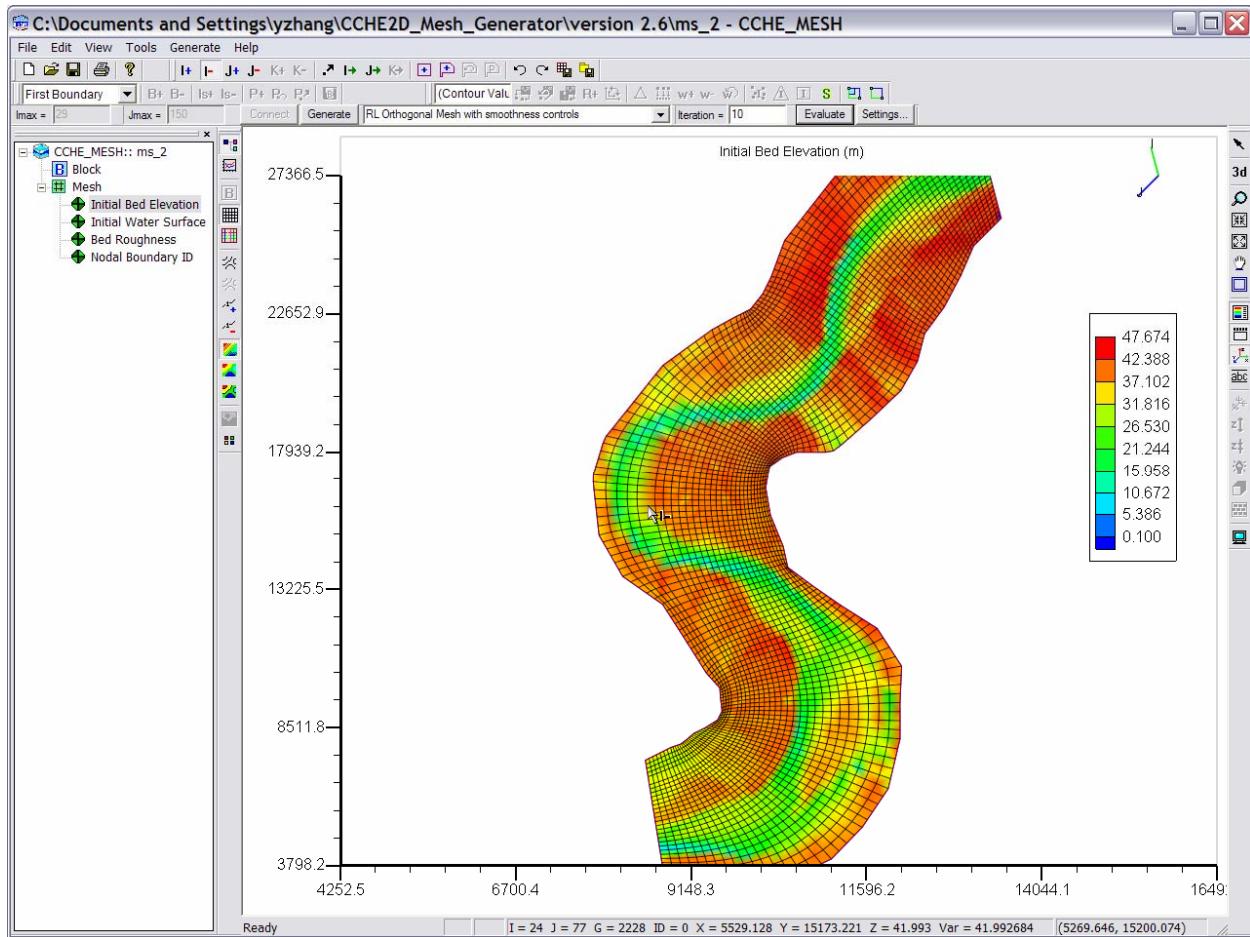


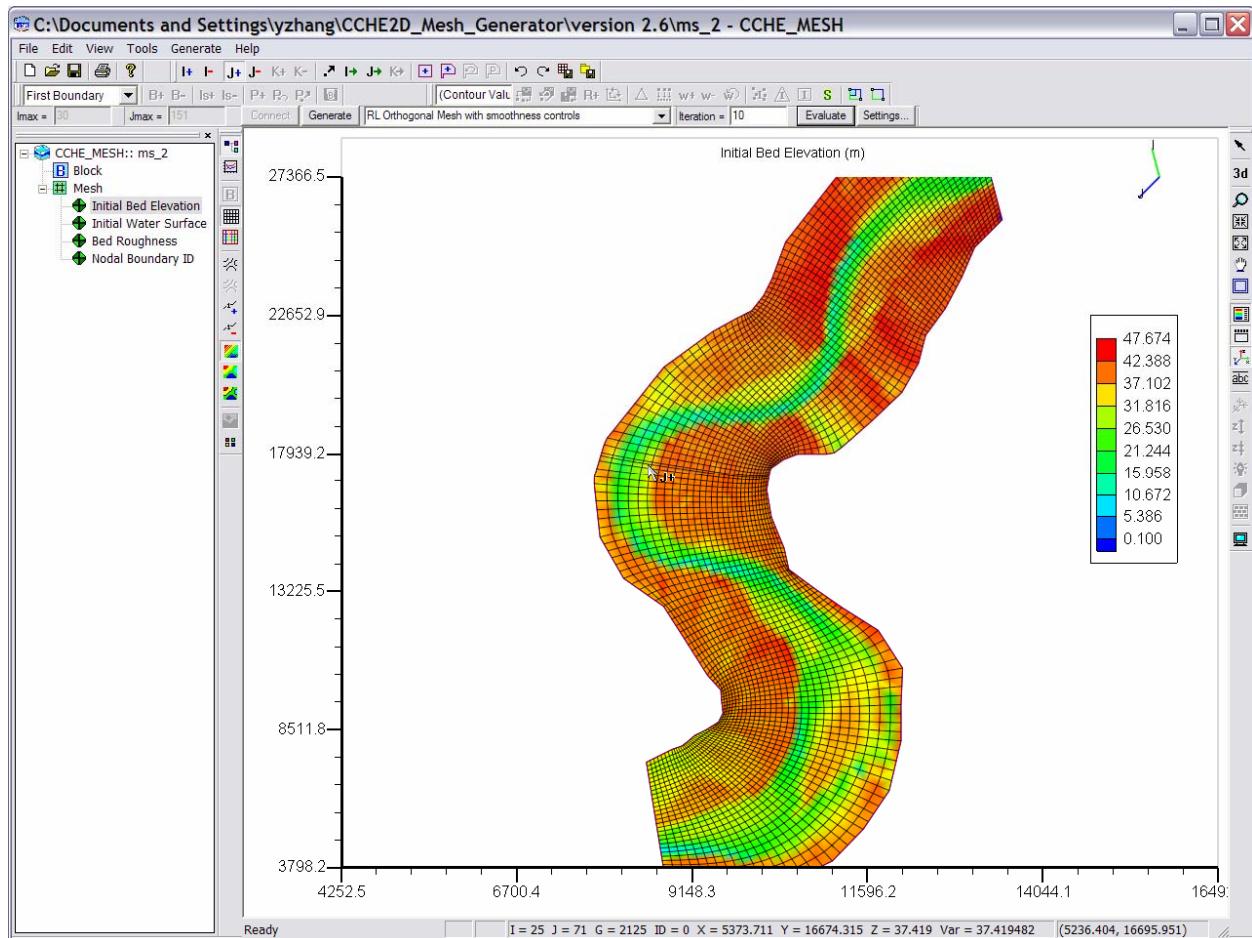
To **Add I/J Line**, click **I+** or **J+** first, then left click the desired place (actually somewhere in between two I/J Lines). A I/J Line will be added at the place you clicked.

To **Delete I/J Line**, click **I-** or **J-** first, then left click the I/J line you want to delete. The I/J line you selected will be deleted. **Note** that you cannot delete I/J line cross the hydraulic structures, i.e., islands, dikes, etc.

To **Move I/J Line**, click **I+** or **J+** first, then left click the I/J line you want to move and hold it, and the release it at the desired location. **NOTE:** you can only move mesh line between two lines..







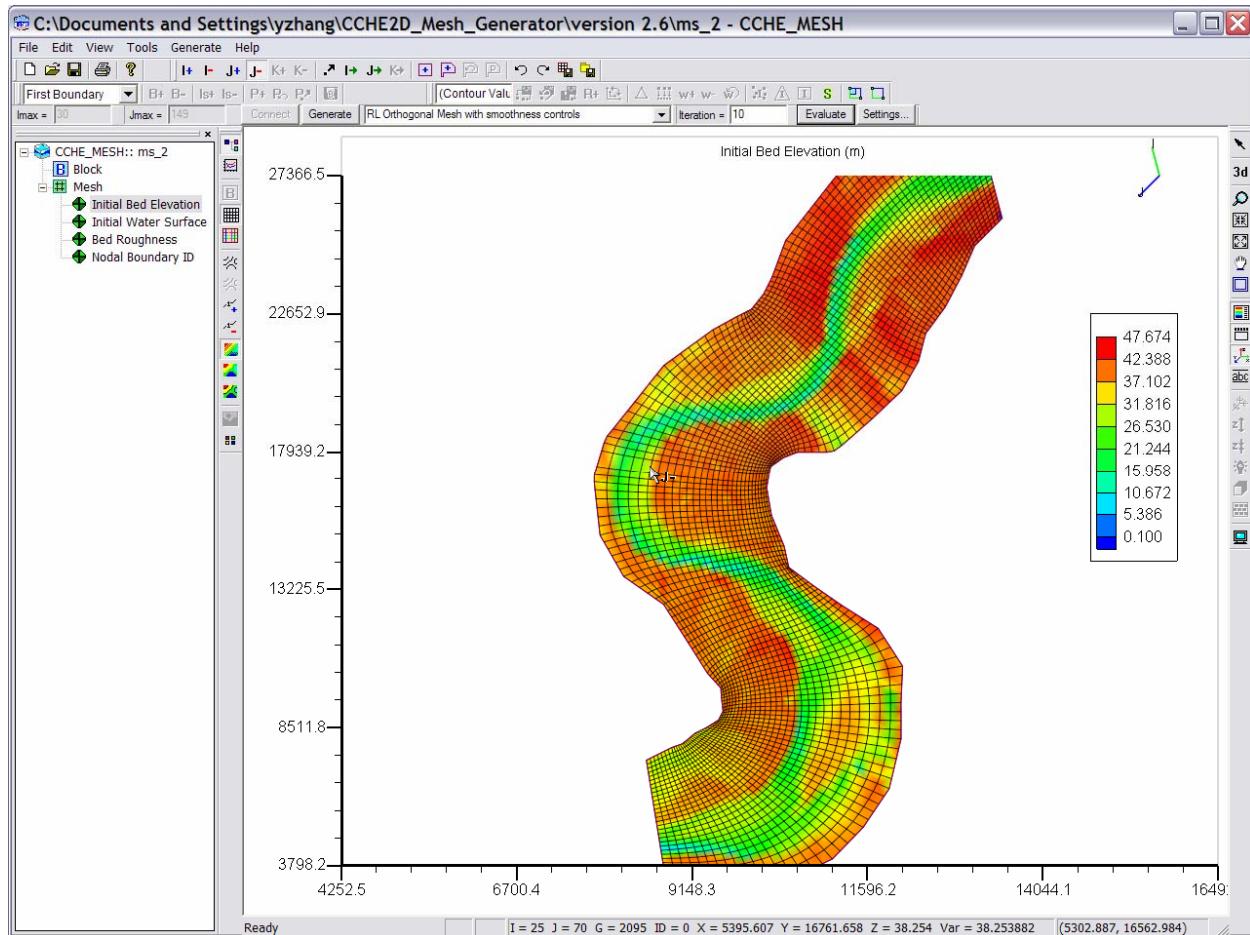
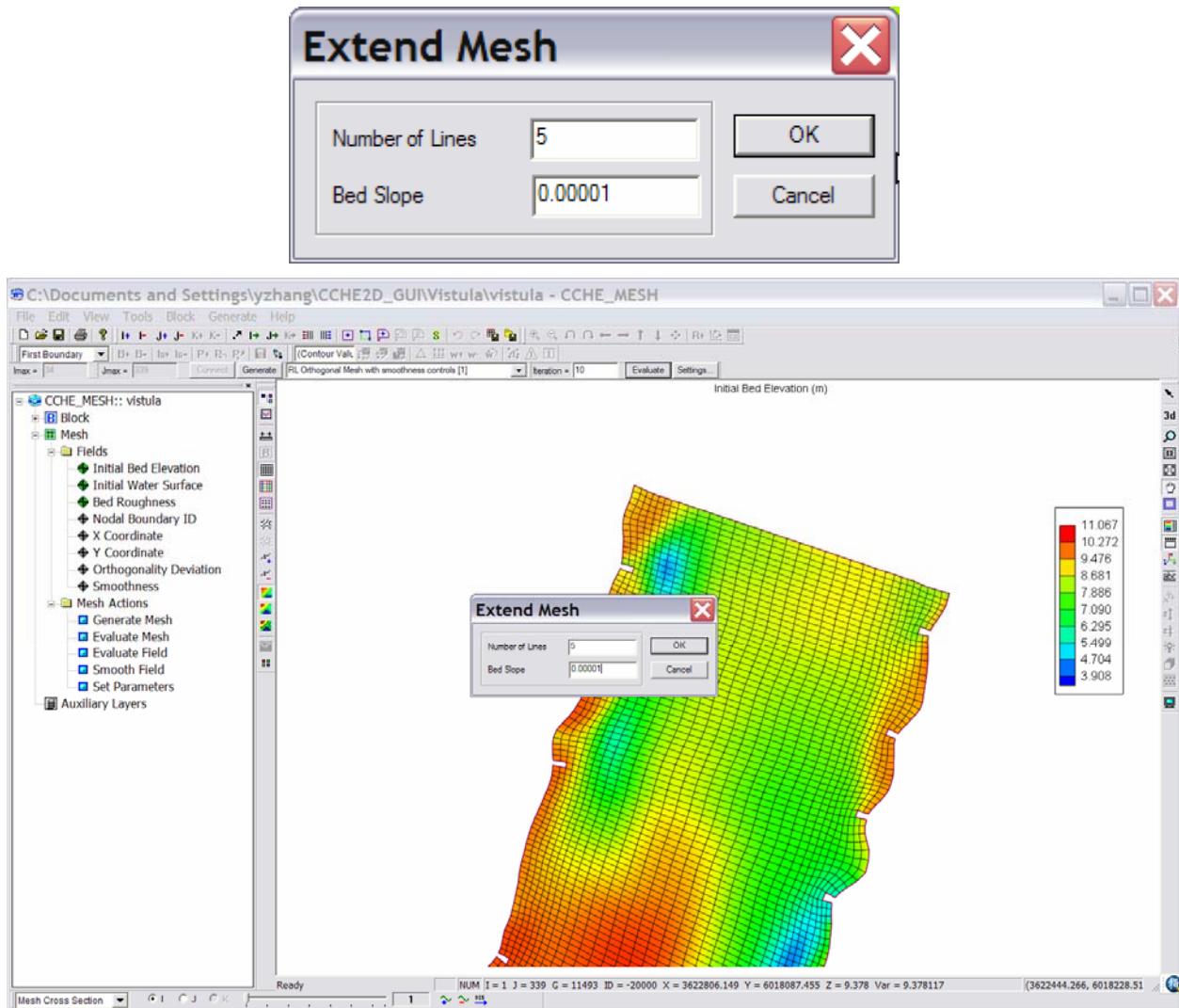


Figure 4-66

To **Move Mesh Node**, click first, then left click the mesh node you want to move and hold it, and then drag it to the desired place and release it.

To **Extend mesh from starting/ending J Line**,

- Click or first;
- In the Extend Mesh dialog window, set the **Number of Lines** to be extended and the **Bed Slope**. Select **OK** and then the specified number of J lines will be extended from the starting or ending J line. The extended J Lines will have the same shape of the starting or ending J line and the bed elevation will be calculated according to the **Bed Slope**.



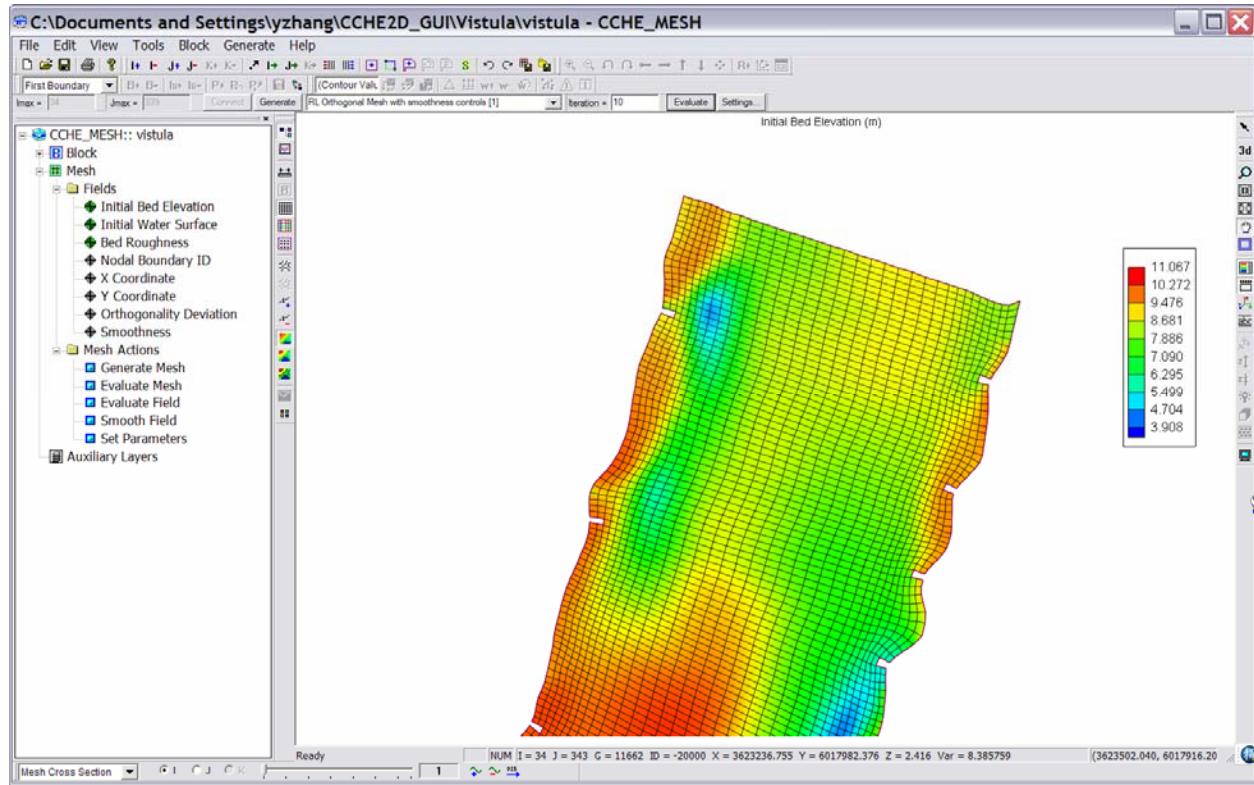


Figure 4-67

4.7.2 Edit Field and Nodal ID

To Change Field Value,

- First select the variable from Project view.

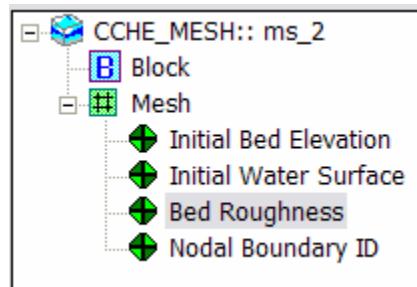
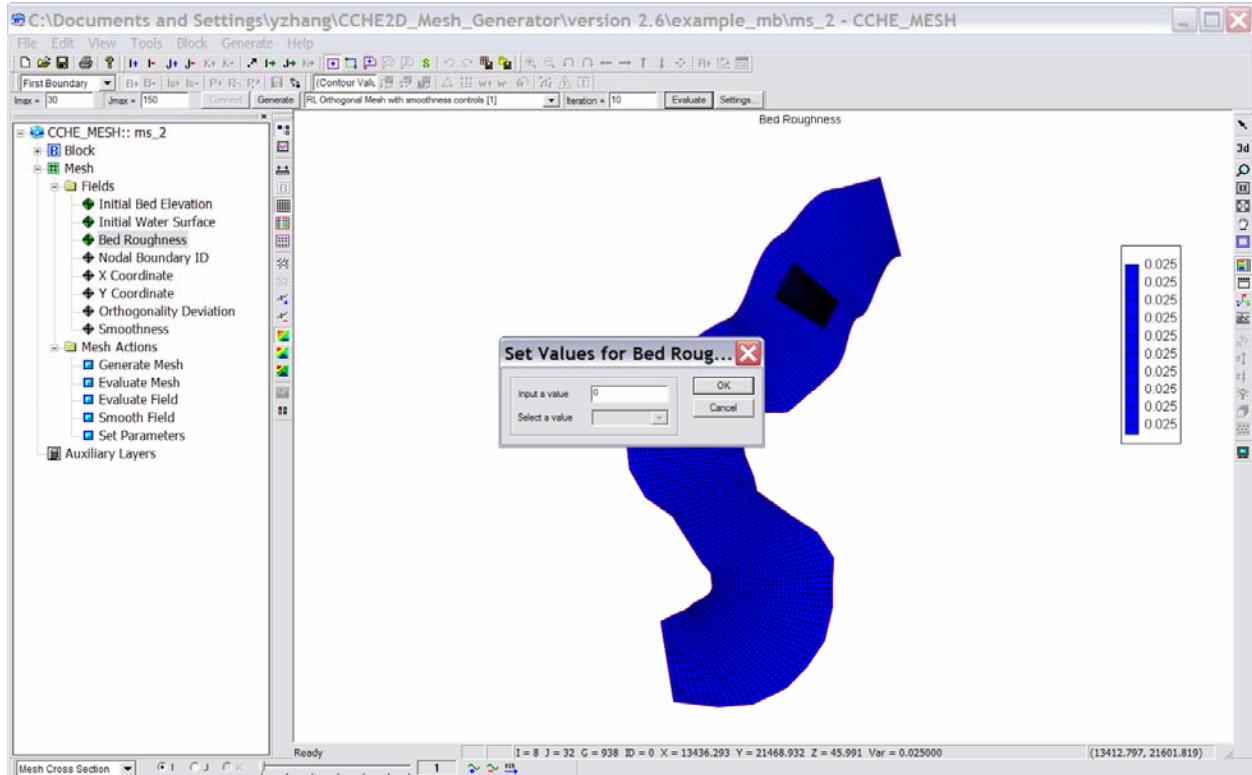


Figure 4-68

- **Define a local area:** you can define either a rectangular area or polygon area. To **define a rectangular area**, select  first and then click two different points on the mesh to define a rectangular area; and, to **define a polygon area**, select  first and then add polygon points by clicking on the desired places on the mesh, and finally **Double-Click the last point** to close the polygon.



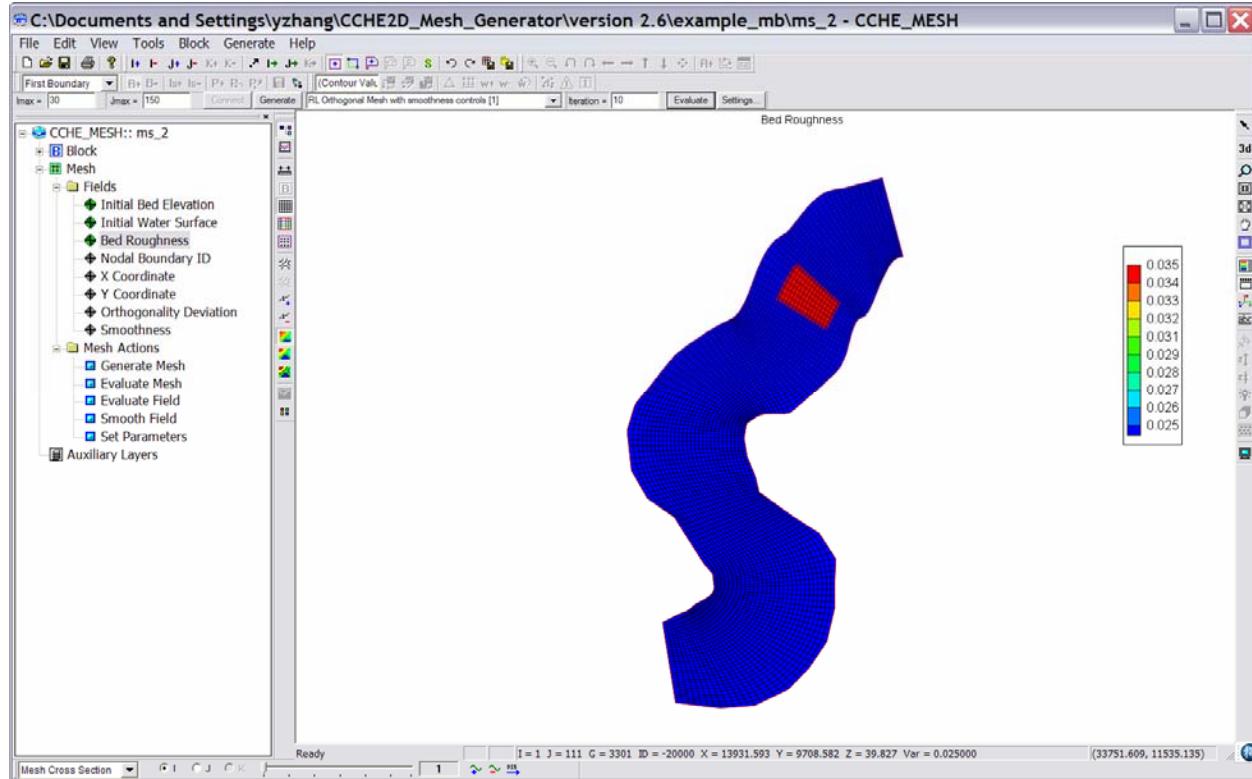


Figure 4-69

- **Set Value:** once finishing defining the local area, you need to set the new value for the defined area.

There are three kinds of nodes for a mesh: internal node ($ID = 0$), boundary node ($ID = -20000$), and external node ($ID = -10000$). The internal node is within the domain; the boundary nodes are those on the boundary; and the external nodes are flagged as outside of domain and will not included for computation.

There are two rules:

- All the internal nodes must be surrounded by the boundary nodes;
- The external nodes can be neighbored only with the boundary nodes.

WARNING: please be careful when changing the nodal ID. If they are not correctly set, the mesh may become invalid and cannot be identified by CCHE2D model.

To Change Nodal Boundary ID,

- First select **Nodal Boundary ID** from **Project** view.
- **Define a rectangular area:** you can only define a rectangular area.
- In **Set Value** window, select a nodal type.

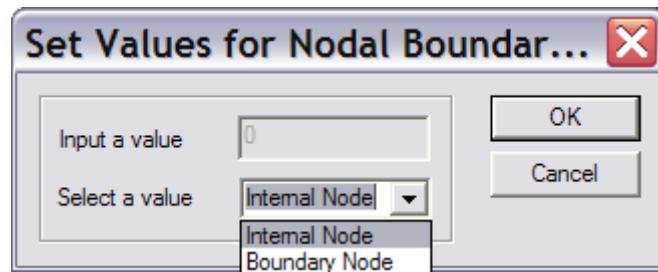


Figure 4-70

To **Undo Previous Change**, click . The CCHE-MESH allows you to undo all your changes step by step.

To **Restore Previous Change**, click . The CCHE-MESH allows you to restore all your changes step by step.

To **Save Changes to the current mesh**, select . If you want to **Save Changes to a new mesh**, select

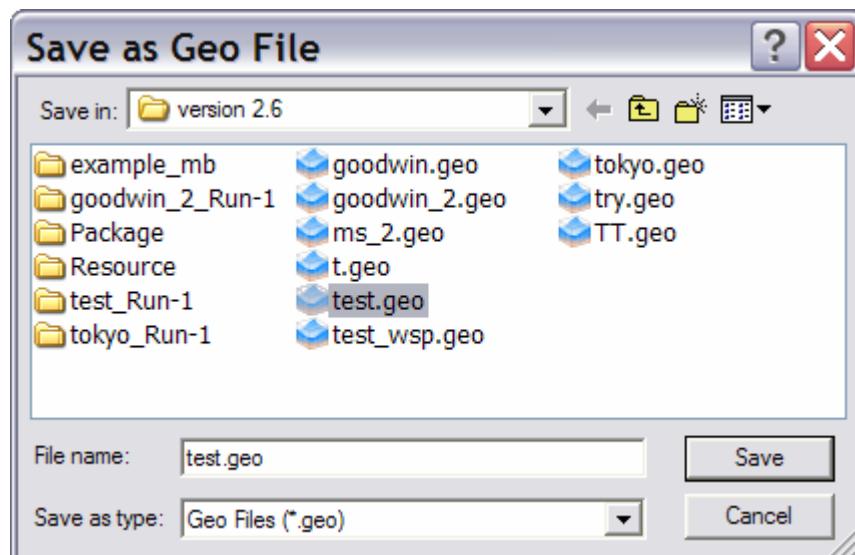


Figure 4-71

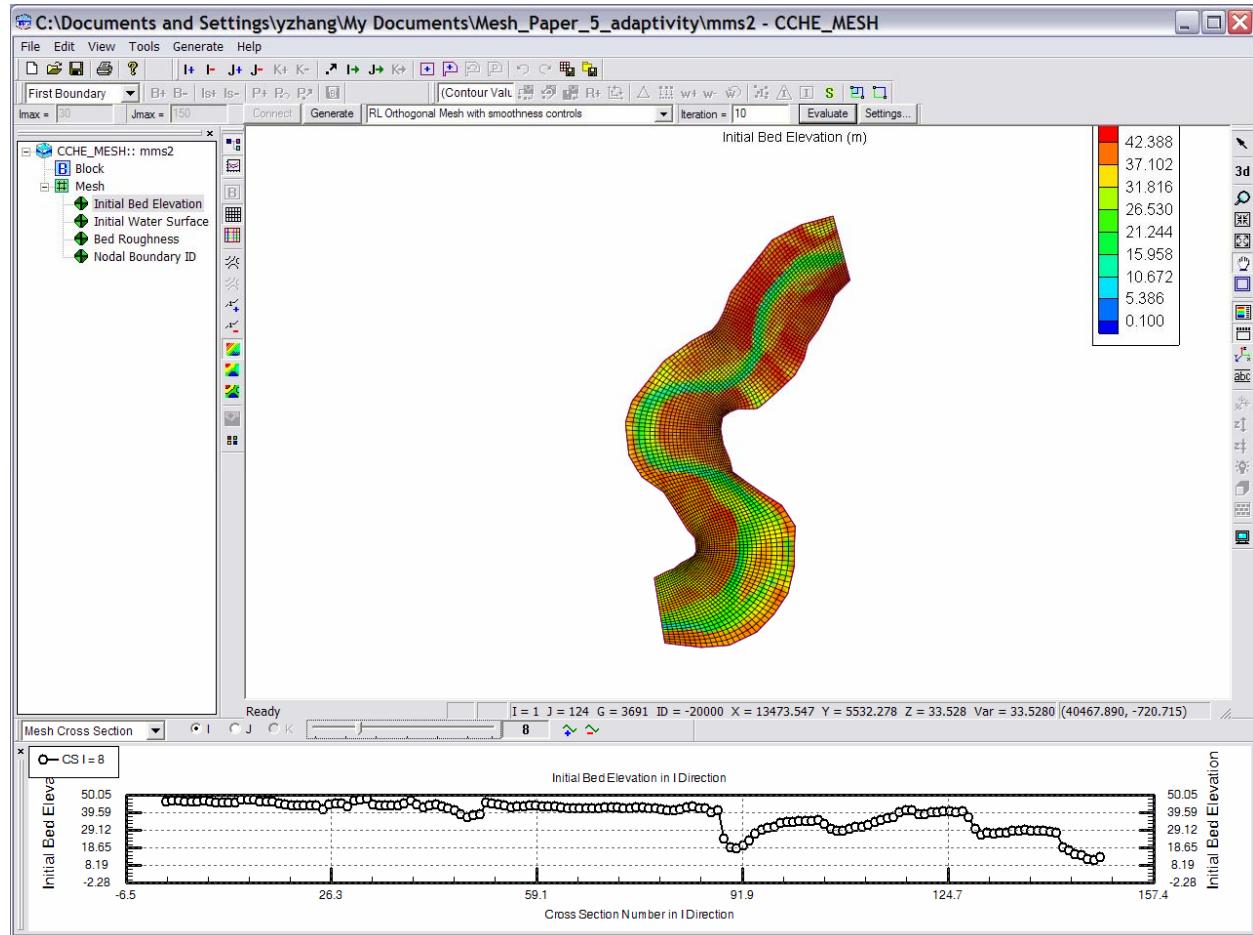
4.8 Visualization

The CCHE-MESH version 3.0 enhanced the visualization capabilities, such as, 2D XY plot and 3D plot.

4.8.1 2D XY Plot

After you load a mesh or a measured cross sections file (*.mesh_mcs) into the CCHE-MESH, you can view the 2D XY Plot.

To activate the XY plot, select  on View toolbar. You can make the **2D XY Plot View** float by **Double-Clicking** the header of the view. The **XY Plot** toolbar will be shown at the same time.



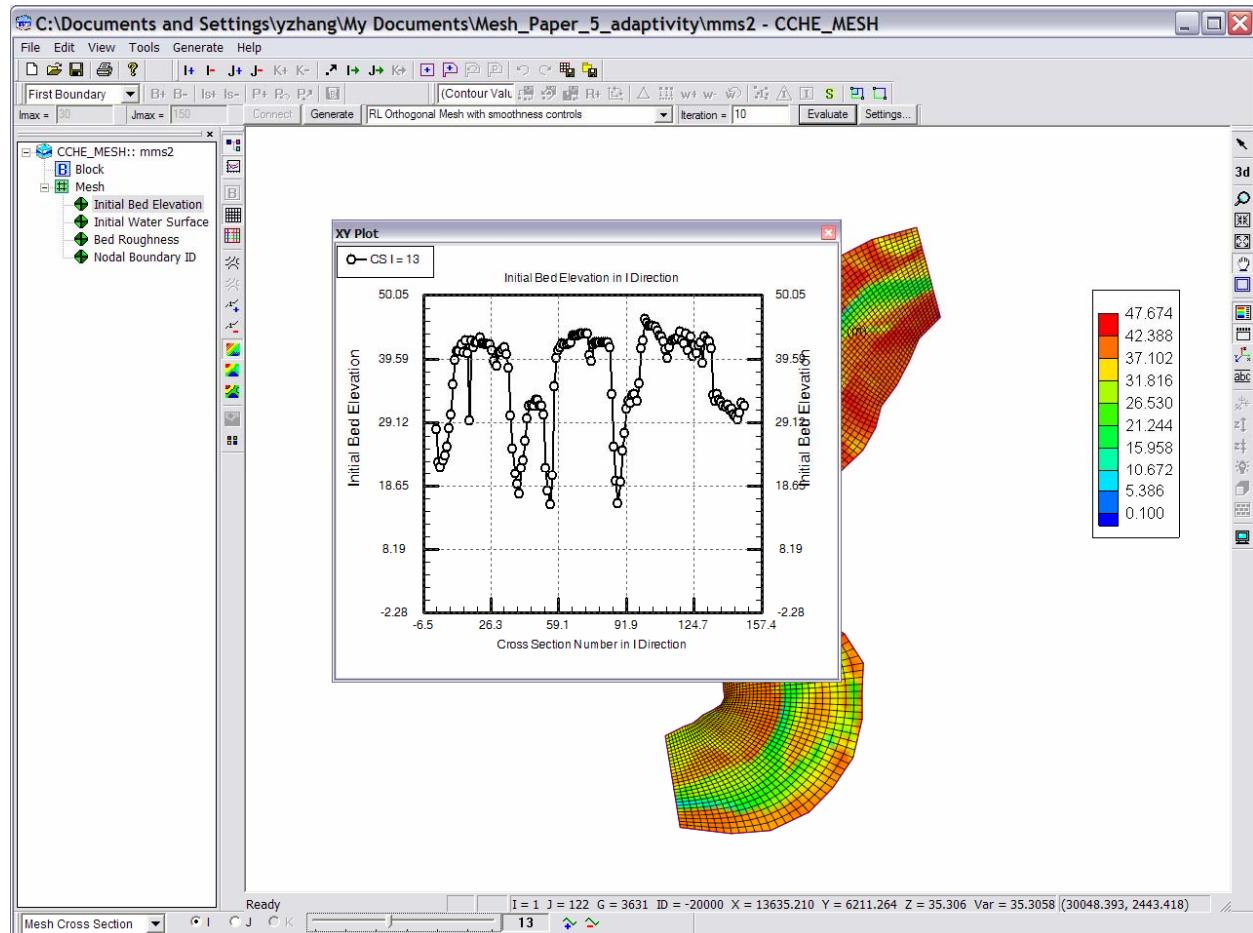


Figure 4-72

In the **XY Plot** toolbar,

- **Mesh Cross Section**: Select the plot type from this selector. There are two kinds of data type: **Mesh Cross Section** and **Measured Cross Sections**.

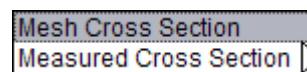


Figure 4-73

- If you load a measured cross section file (*.mesh_mcs), the option **Measured Cross Section** will be selected automatically.

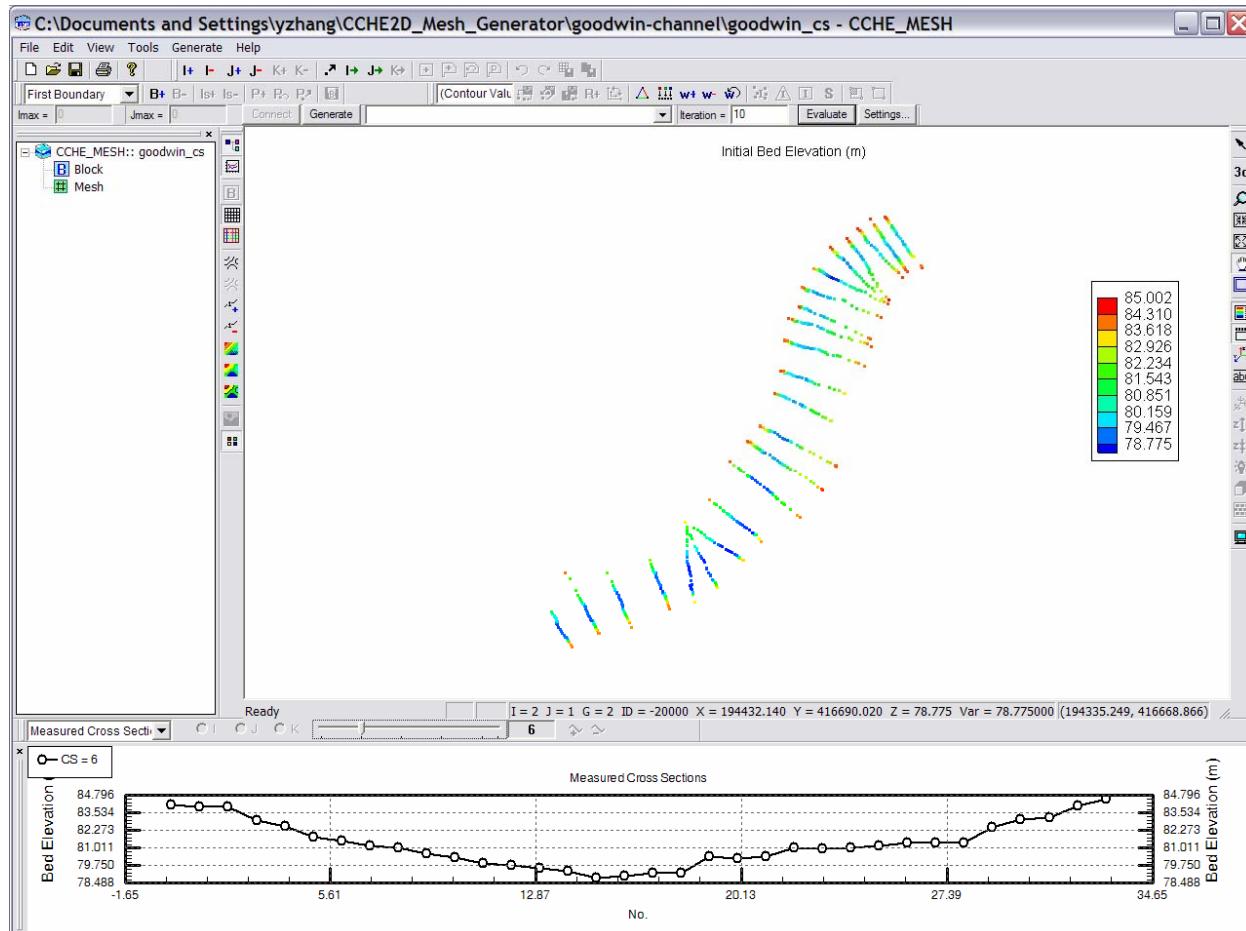


Figure 4-74

- If the option **Mesh Cross Section** is selected, the distribution of **the selected variable** in I direction or J direction will be displayed.
 - You can use the slider to change the I cross section or J cross section.
 - You can view multiple cross sections at the same time. To **add** a cross section, first slide to the desired location, then click ; and similarly, to **delete** a cross section, slide to the desired location and then click .

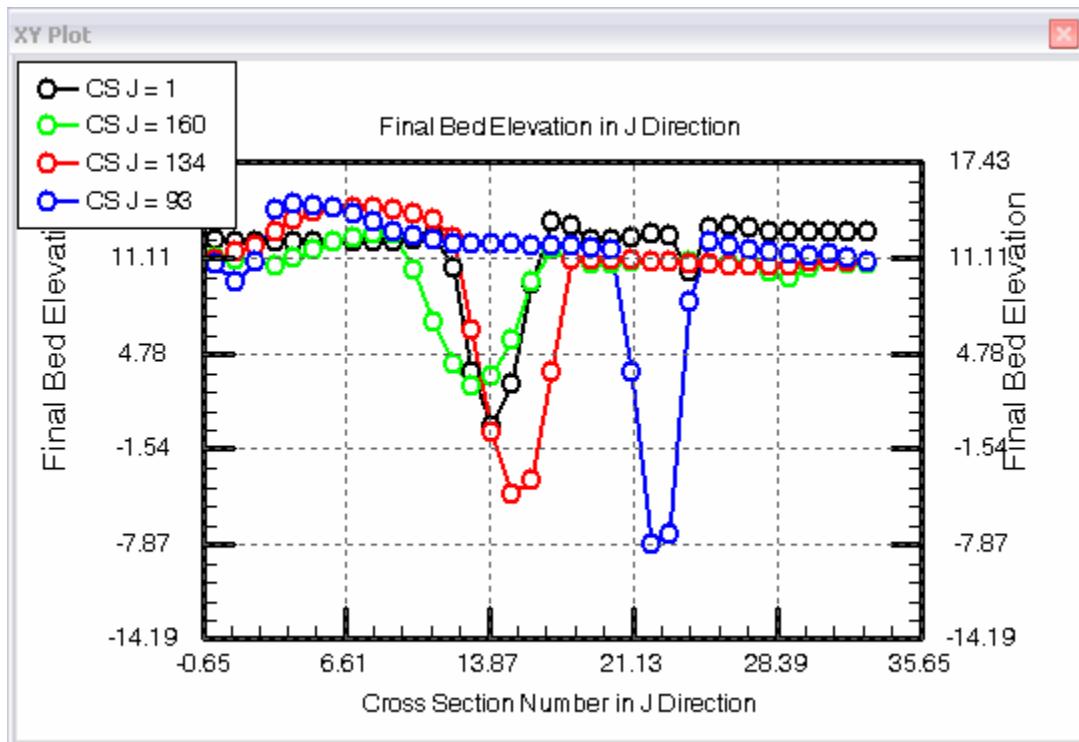


Figure 4-75

- To export the current plot to a bitmap image, first **Right-Click** the XY plot, then in the popup menu choose **Export as Image....**

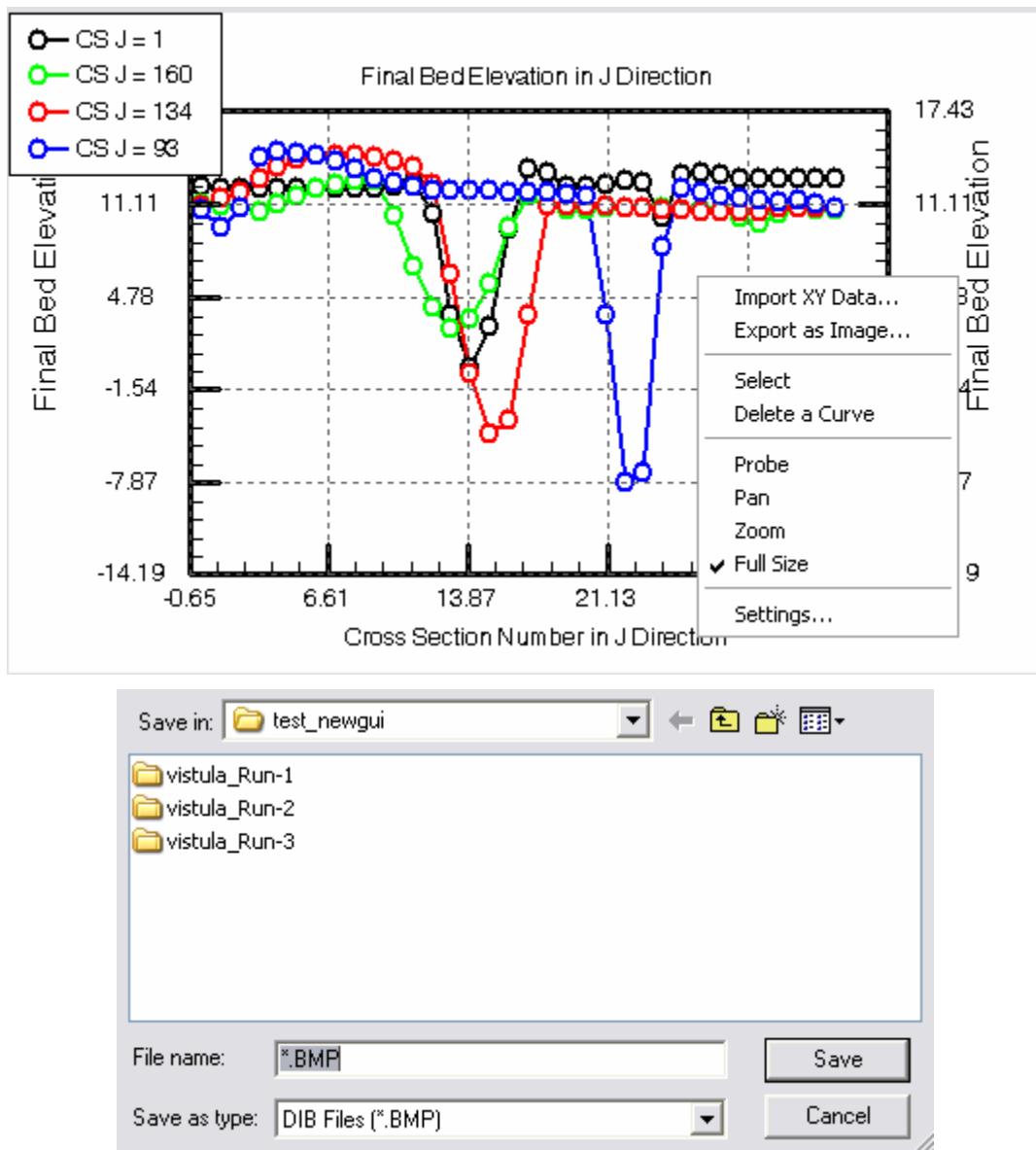


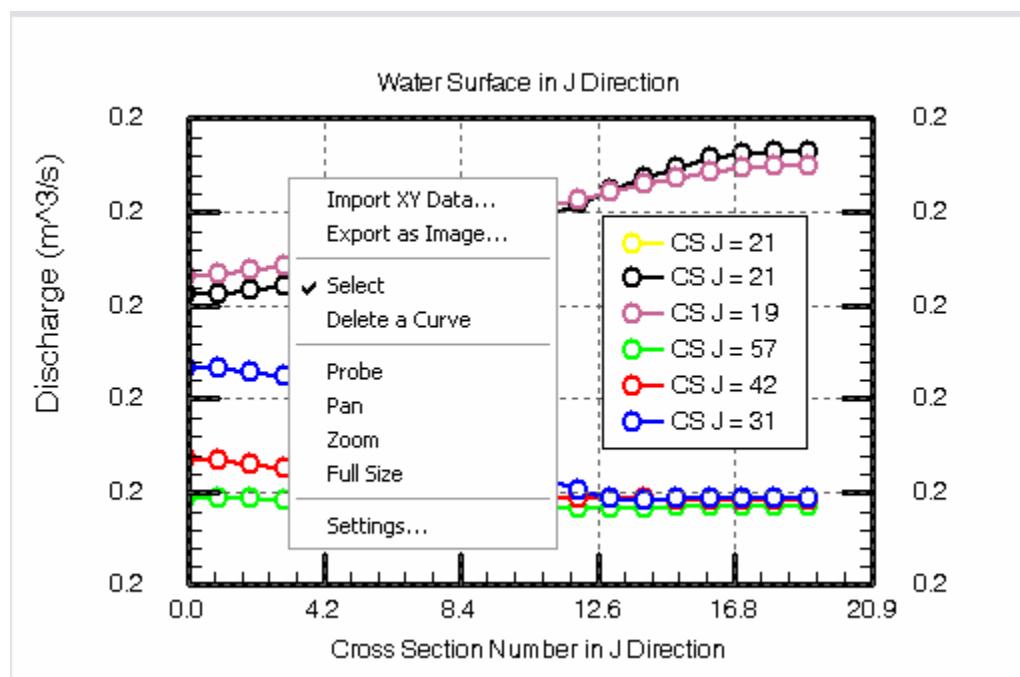
Figure 4-76

The **XY plot** can also show the user-defined data. For example, the XY plot can import the data as shown in Figure 4-77.

To import the user-defined data, first **Right-Click** the **XY plot**, then in the popup menu choose **Import XY Data....**

```
5          //number of curvers
6          //number of points
0.0      10.
2.0      8.0
4.0      5.0
7.0      6.0
12.0     11.0
15.0     7.0
3          //number of points
-2.0     8.0
0.0      3.0
9.0      5.0
5          //number of points
0.0      4.0
6.0      8.0
9.0      5.0
18.0     9.0
20.0     7.0
4          //number of points
1.0      7.0
5.0      6.0
8.0      10.0
13.0     12.0
3          //number of points
2.0      100.0
7.0      120.0
10.0     110.0
```

Figure 4-77



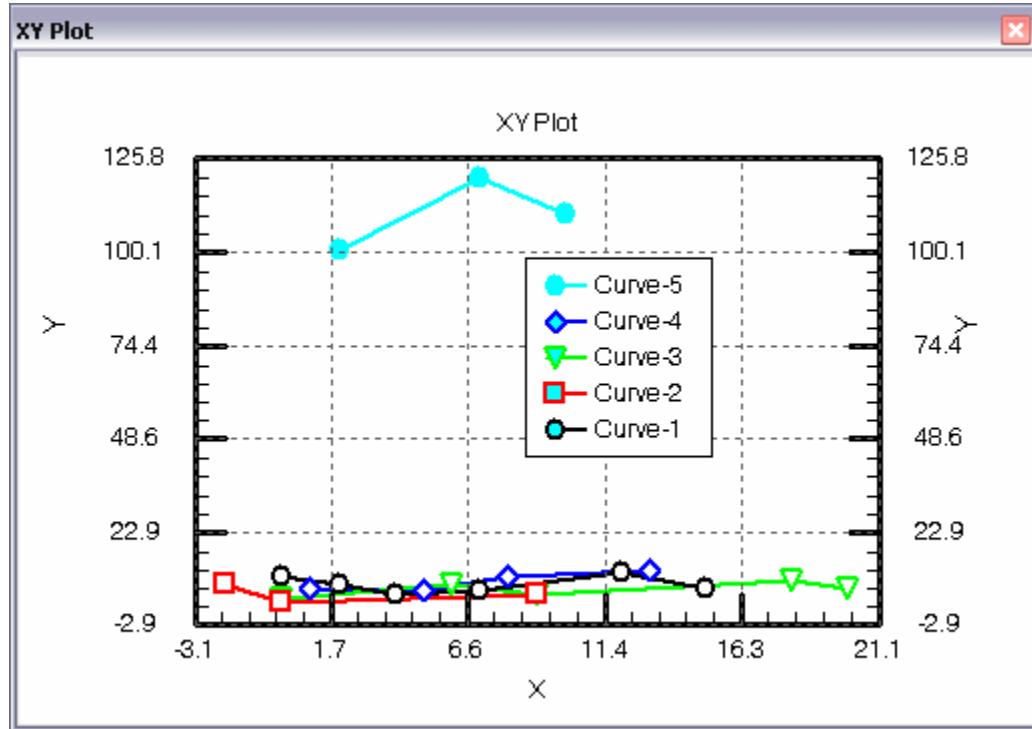


Figure 4-78

You can **Probe**, **Pan**, **Zoom** the plot and set the curve properties. To set the plot properties, first **Right-Click** the plot, then in the popup menu choose **Settings....**

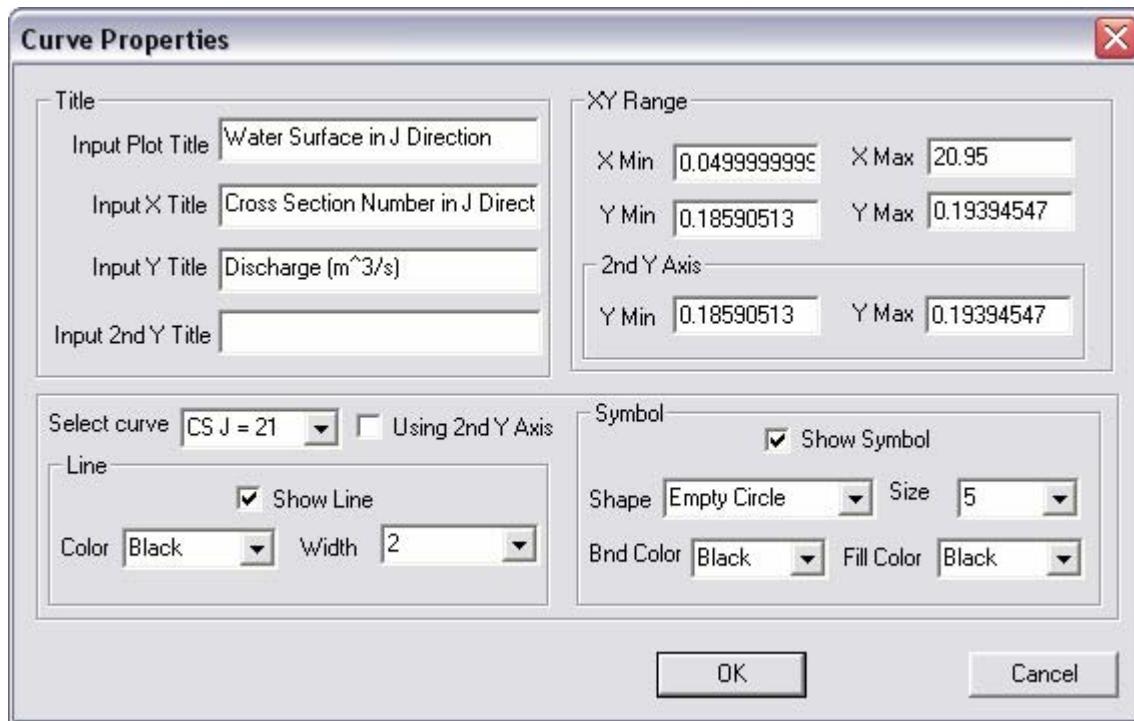


Figure 4-79

4.8.2 View Tools

There are two toolbars related to the graphical view: one is a part of **View** toolbar and the other is **View Tool** toolbar. Please refer to Chapter 3 for details of the functions of them.



Figure 4-80

In the **View** toolbar, you can show/hide **Block**, **Mesh**, **Contours**, **Image** and **Scatter Points**

- **Block** : Select this button to show or hide blocks.

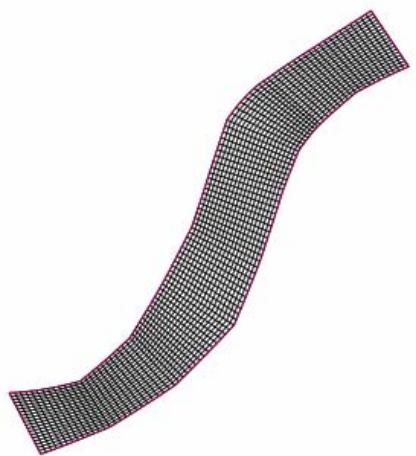
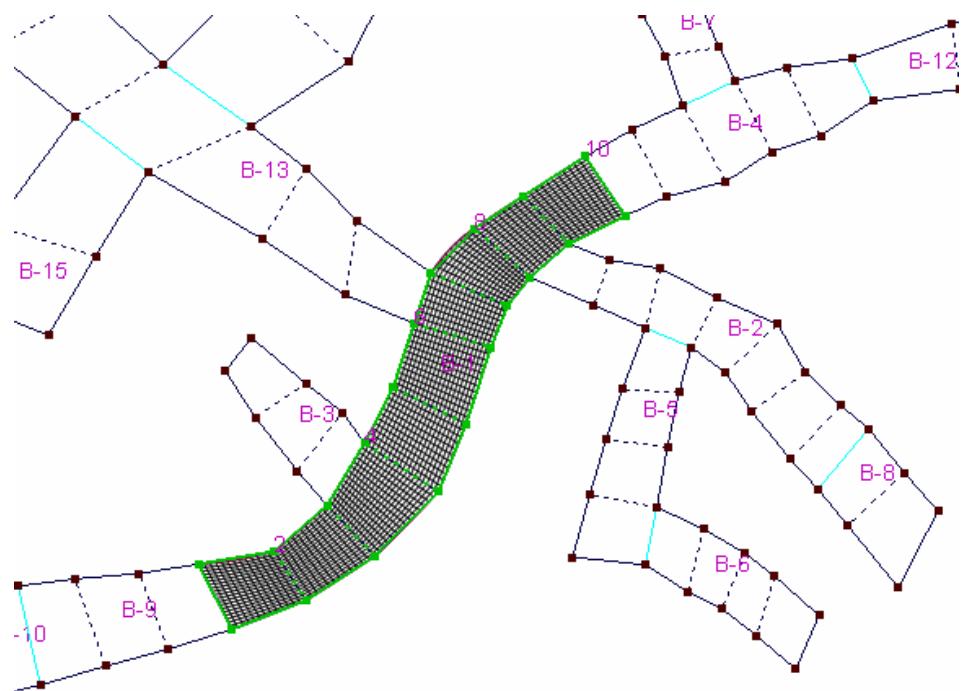


Figure 4-81

- **Mesh** : Select this button to show or hide mesh.

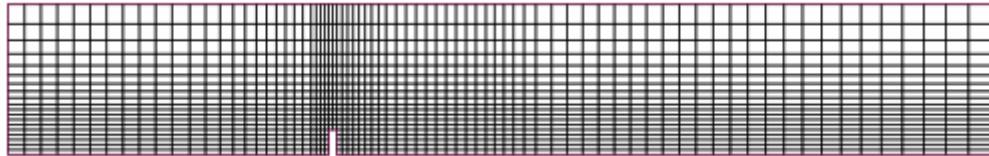


Figure 4-82

- **Colored Mesh** : Select this button to show or hide colored mesh.

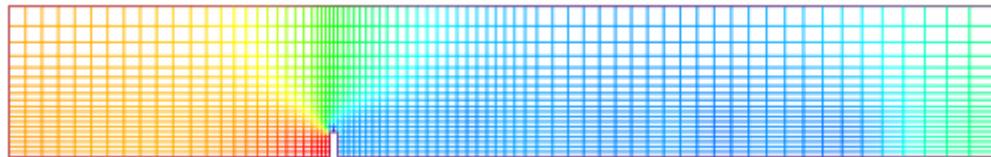


Figure 4-83

- **Contour Line** : Select this button to show or hide contour line.

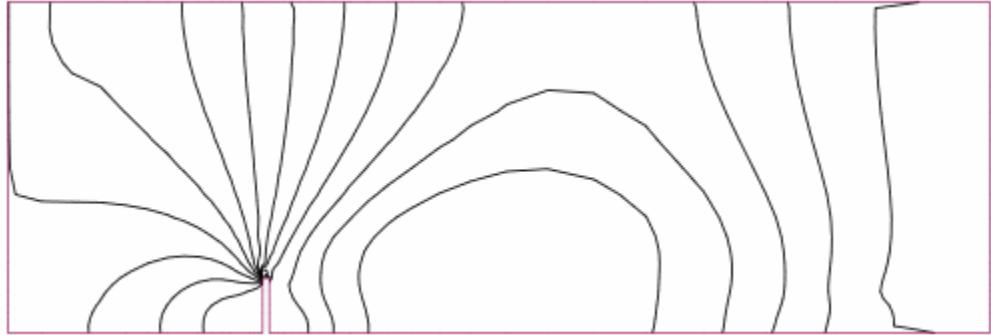


Figure 4-84

- **Colored Contour Line** : Select this button to show or hide colored contour line.

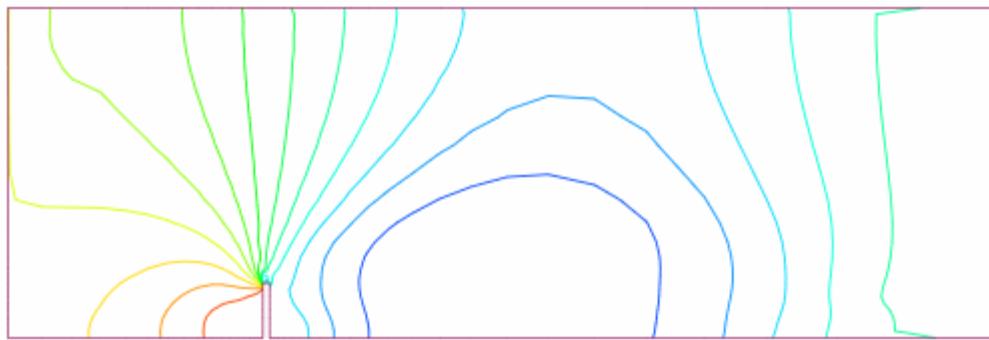


Figure 4-85

- **Add Contour Label** : Select this button to add contour labels. You need to click on a contour line, then the label of this selected contour line will be displayed.

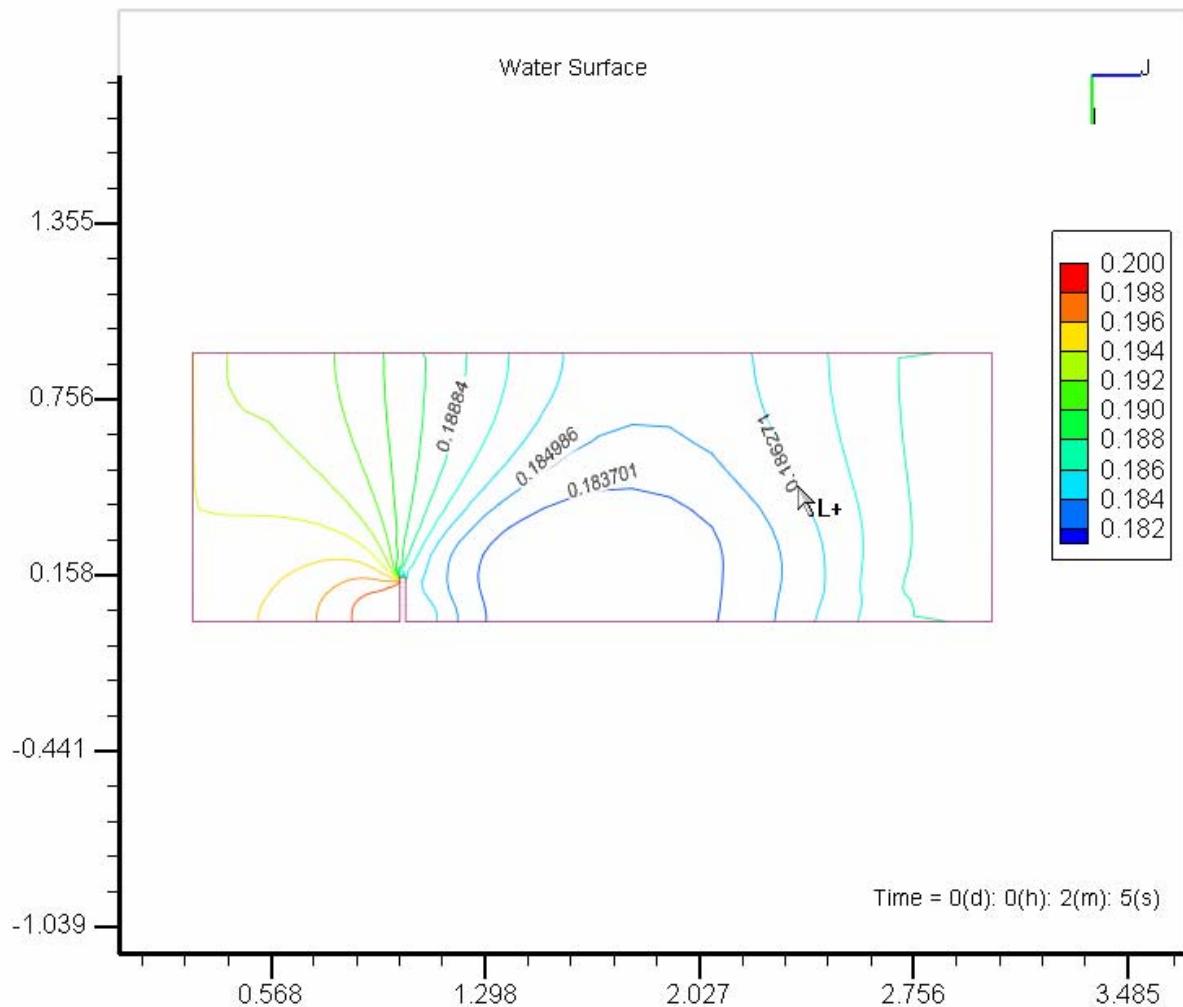


Figure 4-86

- **Delete Contour Label** : Select this button to delete contour label. You need to click the label you want to delete. If you change the variable to be displayed, all the labels will be deleted automatically.

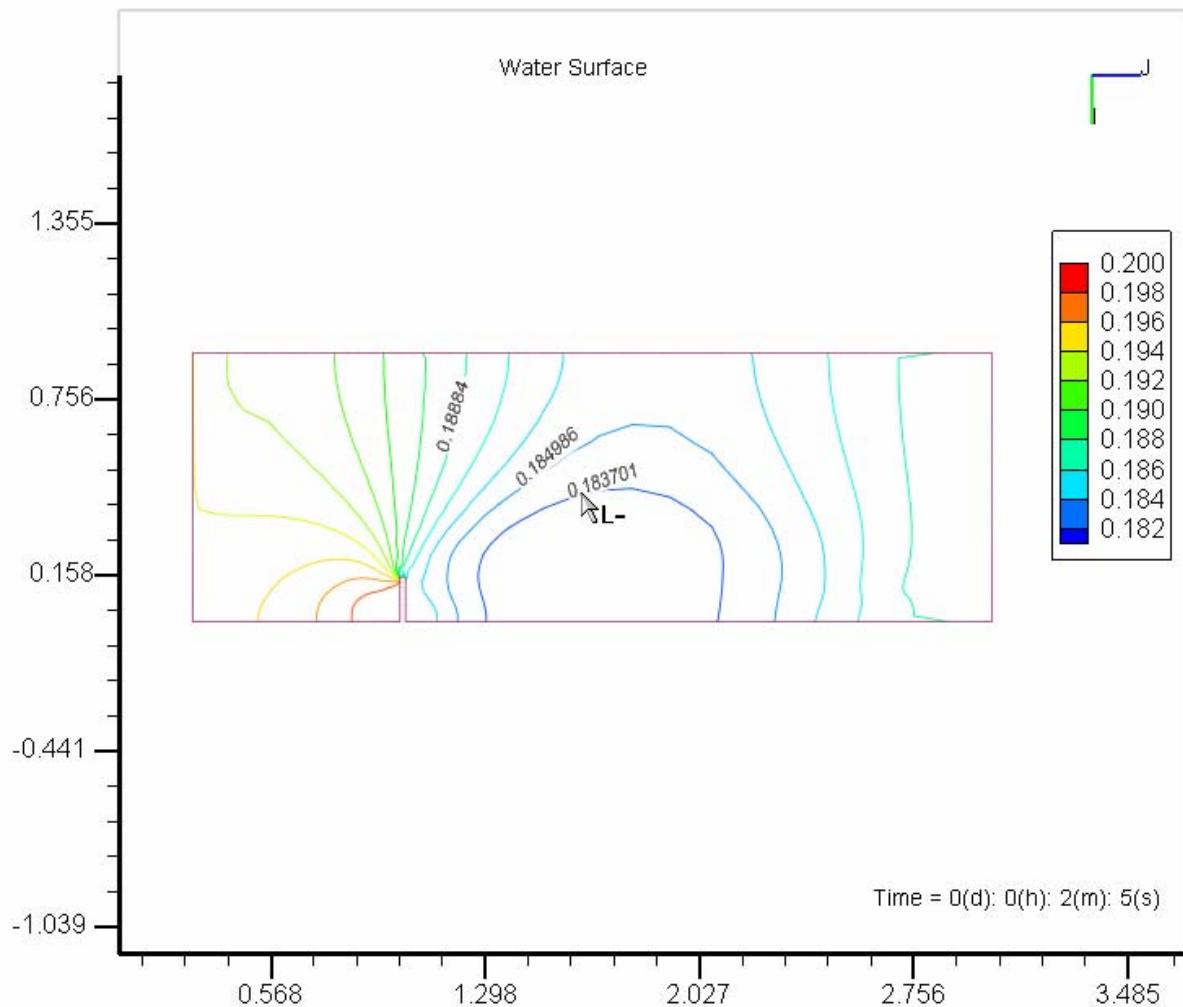


Figure 4-87

- **Flood Shading** : Select this button to show or hide flood shading.

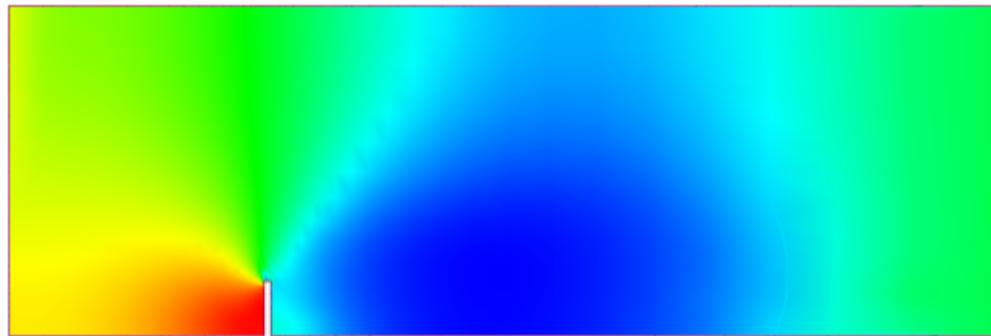


Figure 4-88

- **Contour Shading** : Select this button to show or hide contour shading.

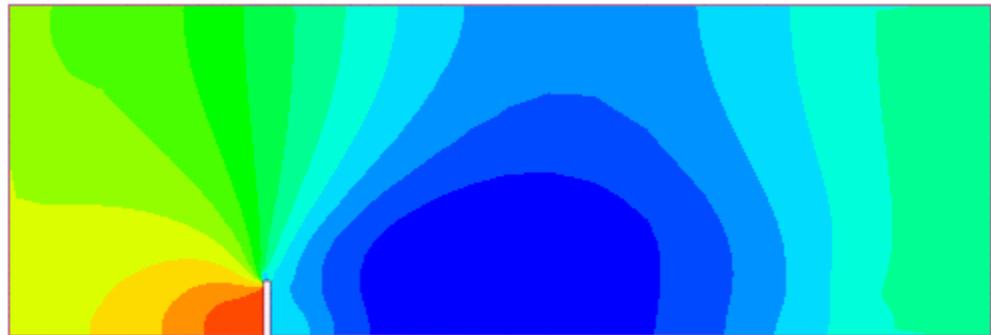


Figure 4-89

- **Contour Shading + Contour Line** : Select this button to show or hide contour with flood + lines.

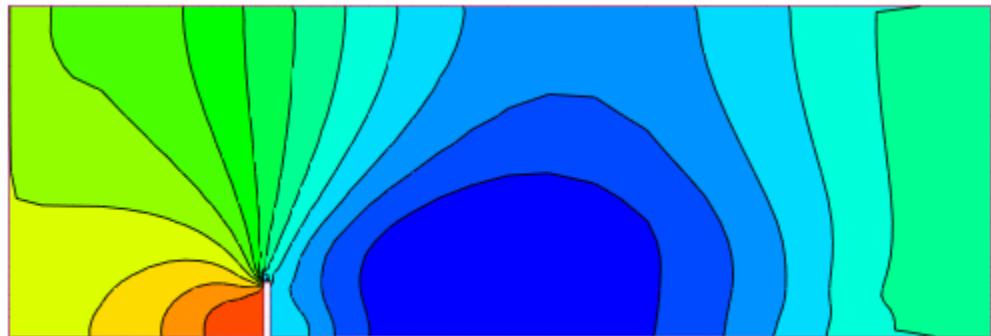


Figure 4-90

- **Image** Select this button to show or hide the image. Please make sure you already imported a bitmap image. **Note:** Only 24-bit bitmap image can be imported. The image editing functions will be covered in later section.

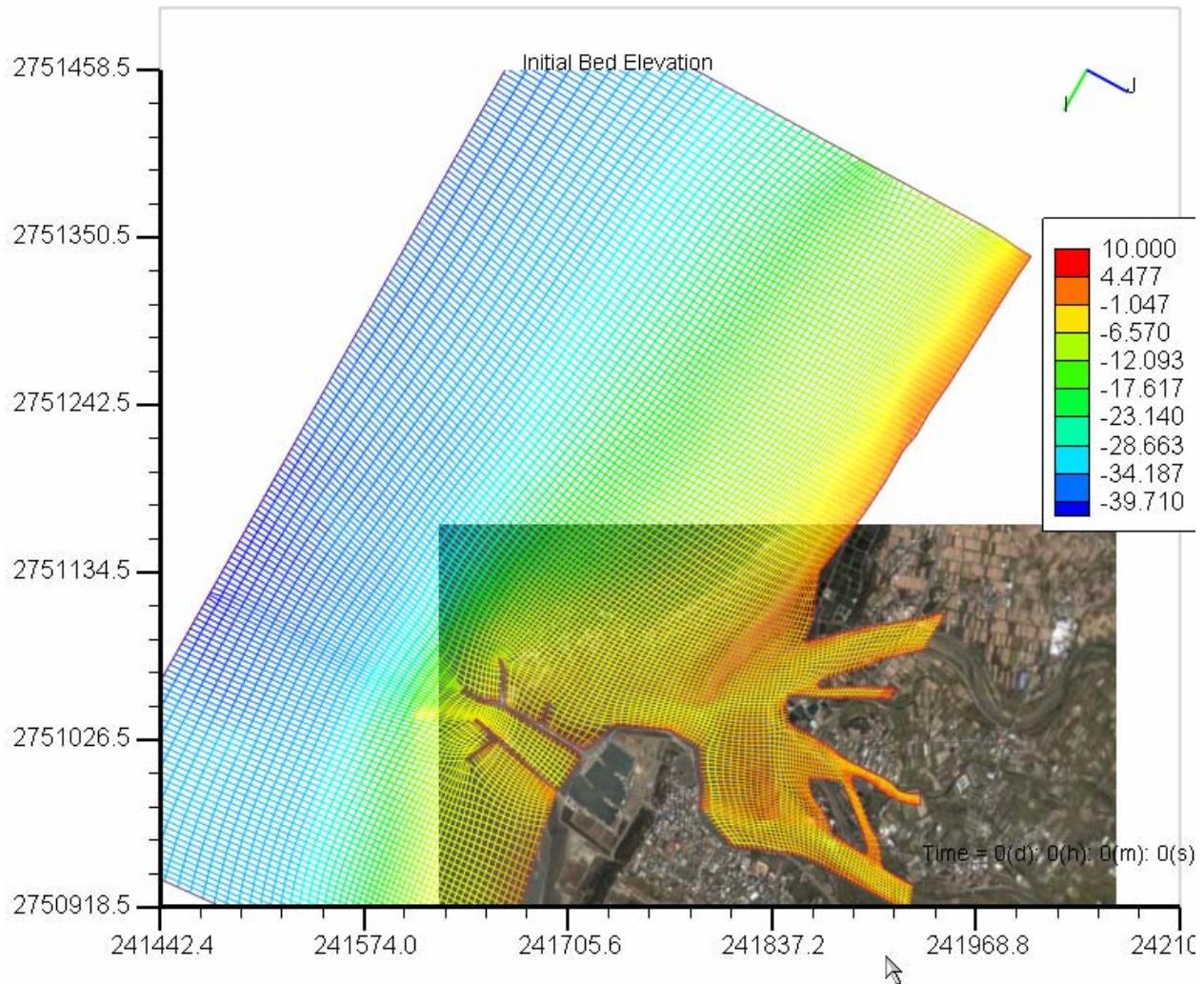


Figure 4-91

- **Scatter Points** Select this button to show or hide the scatter points. Please make sure the scatter points are imported into the CCHE-MESH.

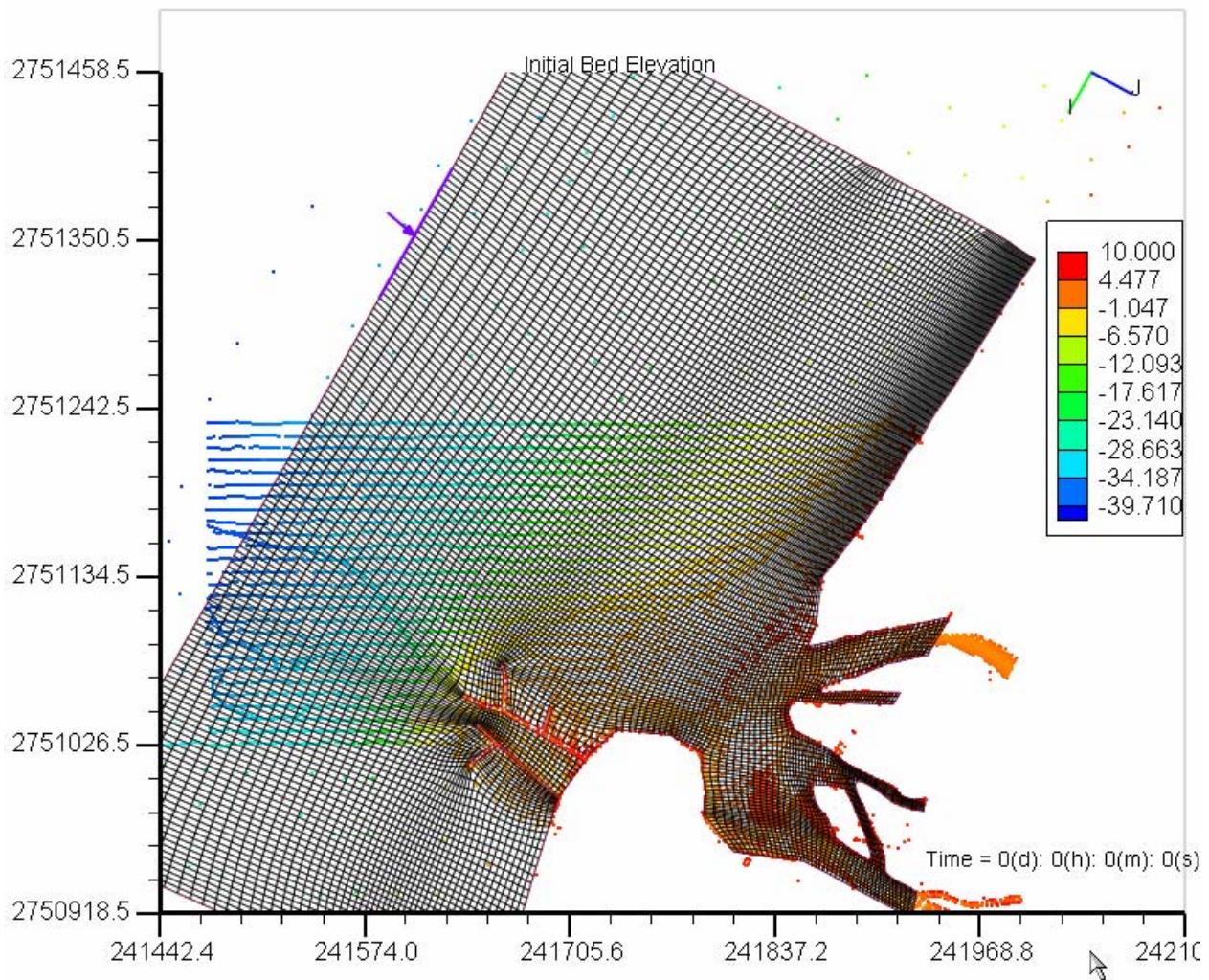


Figure 4-92

In the **View Tool** toolbar, you can view **3D**, **Zoom**, **Pan**, **Rotate** the plot, add **Light**, **Texture** effects.

- **Select** : Select this button to select **Title**, **Text**, and **Legend** and **move** them.
- **3d** : Select this button to show 3D view.

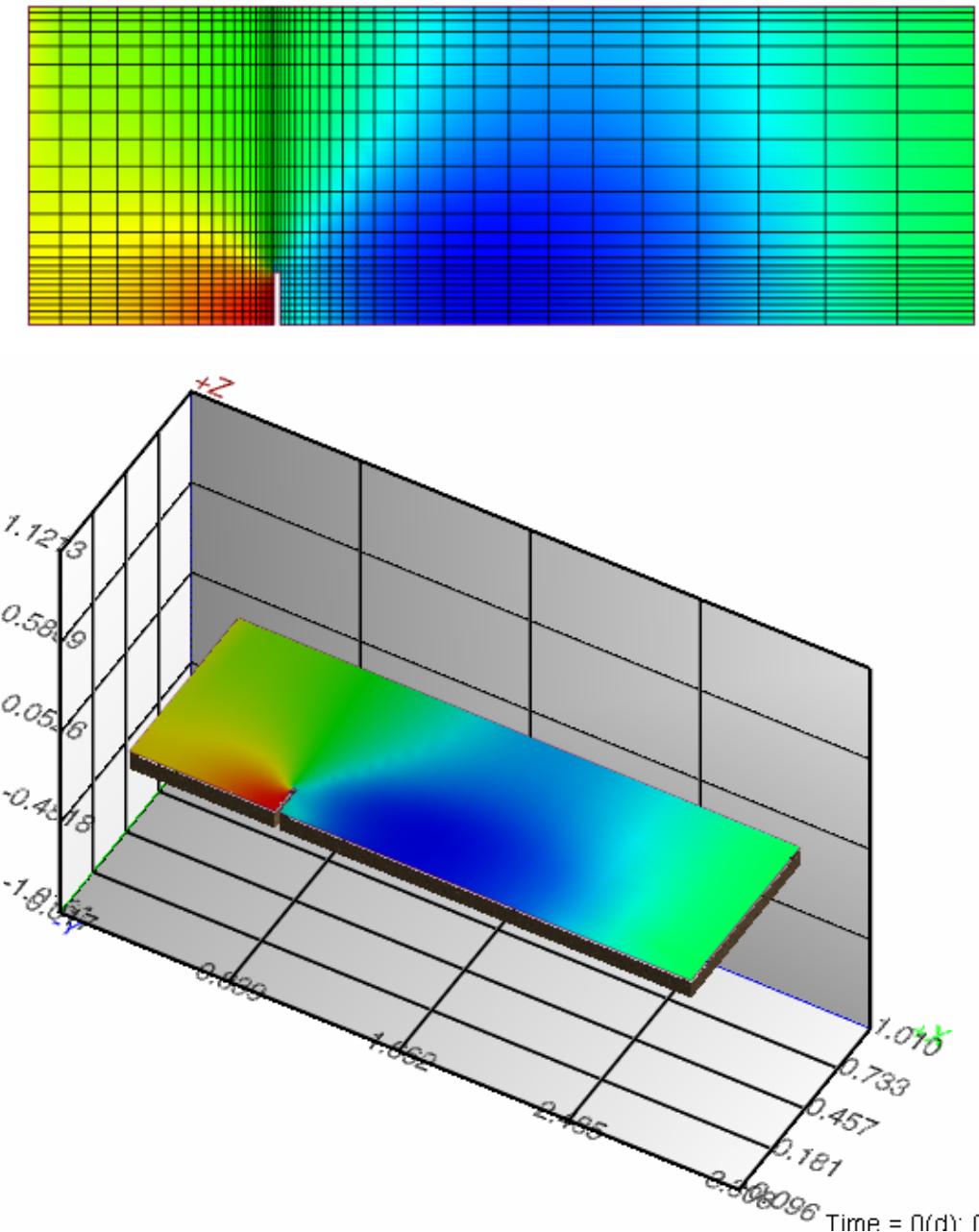


Figure 4-93

- **Zoom** : Select this button to zoom in.
- **Incremental Zoom In** : Select this button to zoom in incrementally.

- **Incremental Zoom Out** : Select this button to zoom out incrementally.
- **Pan** : Select this button to pan.
- **Full Size** : Select this button to restore to full size view.
- **Legend** : Select this button to show or hide legend.

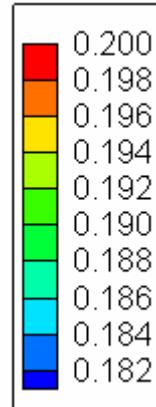


Figure 4-94

- **Title** : Select this button to show or hide title.

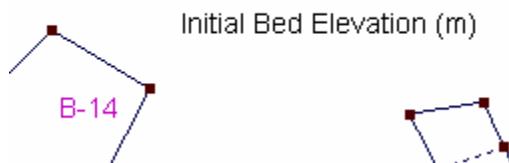


Figure 4-95

- **Axis** : Select this button to show or hide axis.

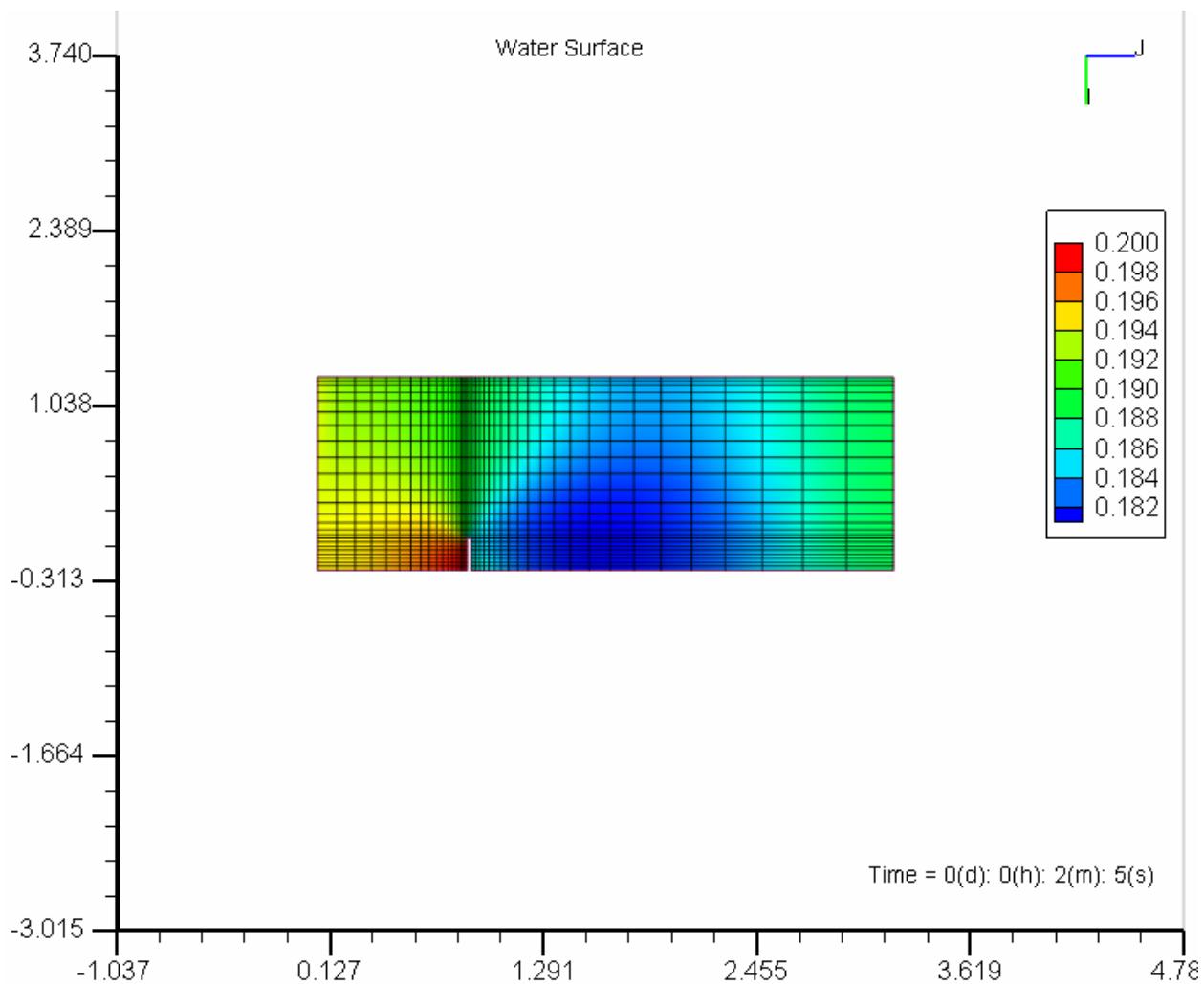


Figure 4-96

- **Text** : Select this button to add the texts.
 - To **add** a text, first select , and then click the desired place where you want to put the text. In the **Add Text** window, input the text and set the text properties. There are two types of texts: 2D and 3D. The former is the flat 2D text and cannot be rotated and the latter is 3D text and can be rotated. If you select the **Local Coordinates System**, the text cannot be moved with the mesh; and, if you select **World Coordinates System**, the text can be moved with the mesh.

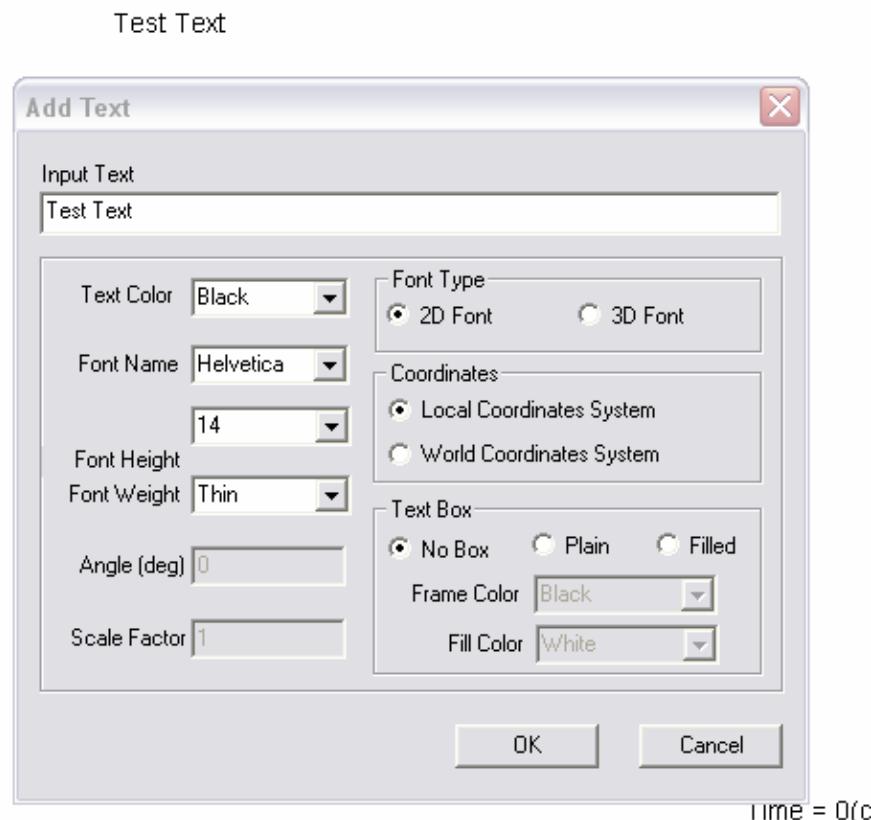


Figure 4-97

- To delete a text, first select , then select the text you want to delete, and then press “Del” key on the keyboard.

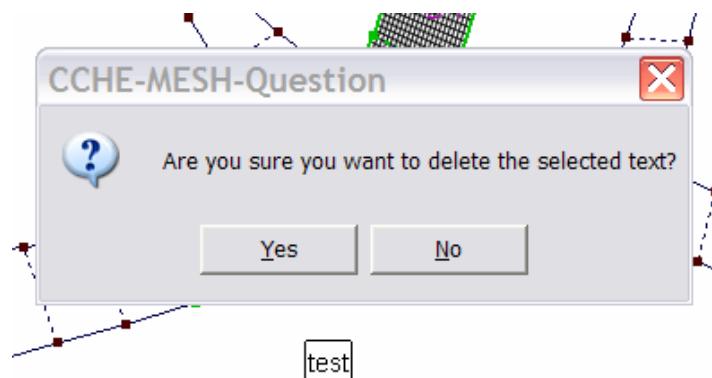


Figure 4-98

- **Rotate** : Select this button to rotate the view using the mouse. When rotating, press the **Left-Button** and **Hold**, and then move the mouse.
- **Increase Z** : Select this button to increase the scale in Z direction.

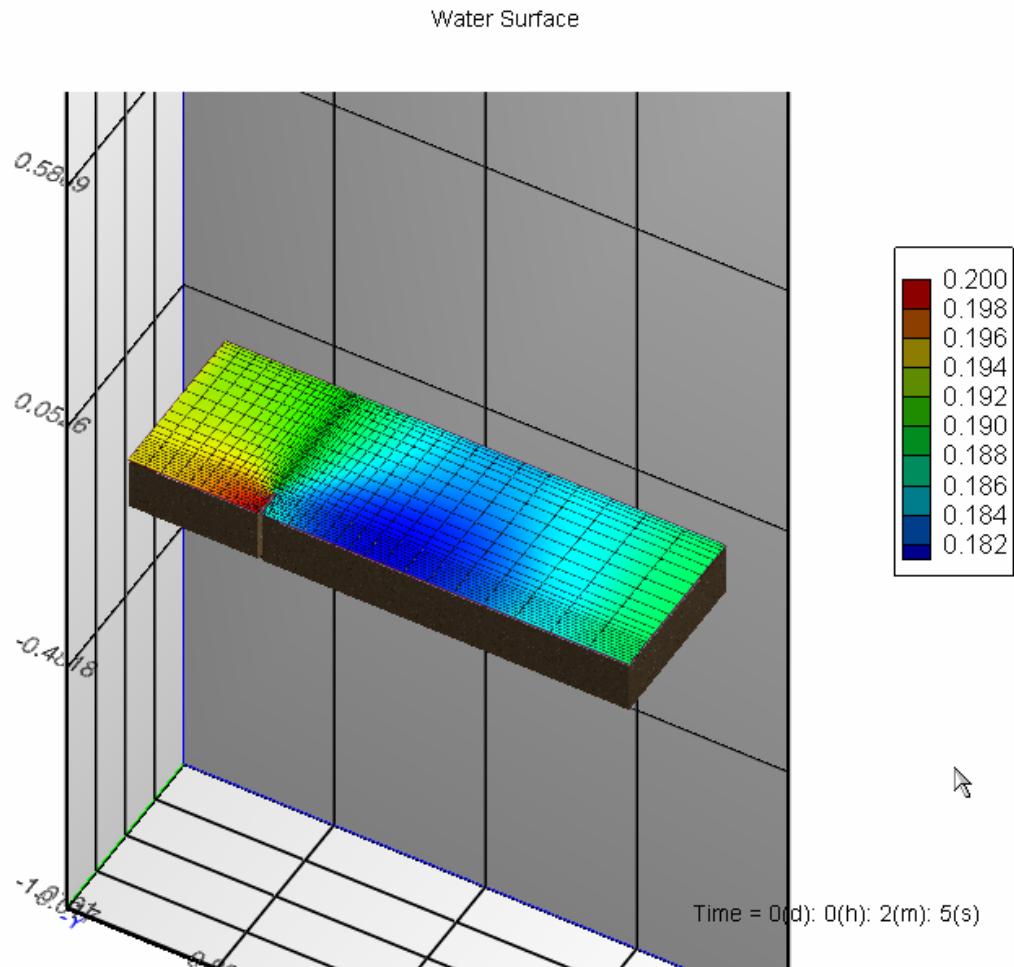


Figure 4-99

- **Decrease Z** : Select this button to decrease the scale in Z direction.

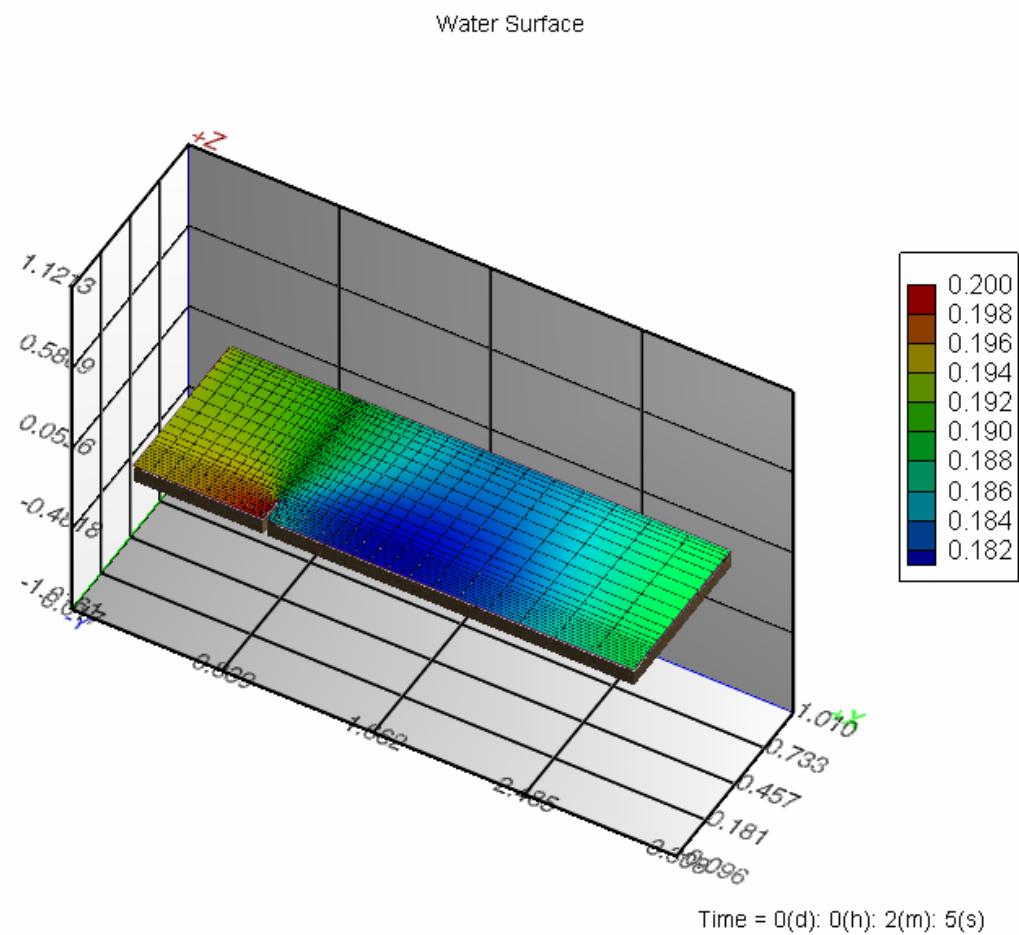


Figure 4-100

- **Light** : Select this button to enable or disable light effects.

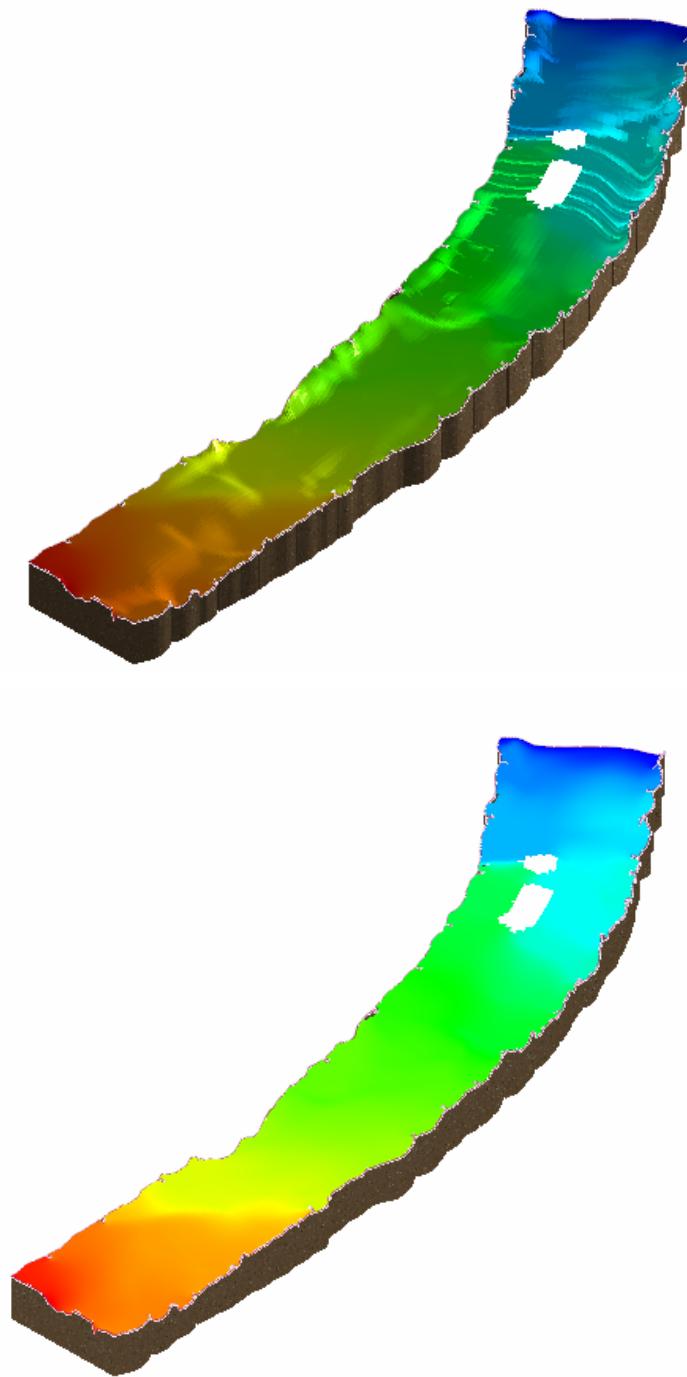


Figure 4-101

- **Frame** : Select this button to show or hide 3D frame.



Figure 4-102

- **Texture** : Select this button to enable or disable texture effects. Figures 5-67 and 5-68 shows the plot with and without texture effects, respectively.

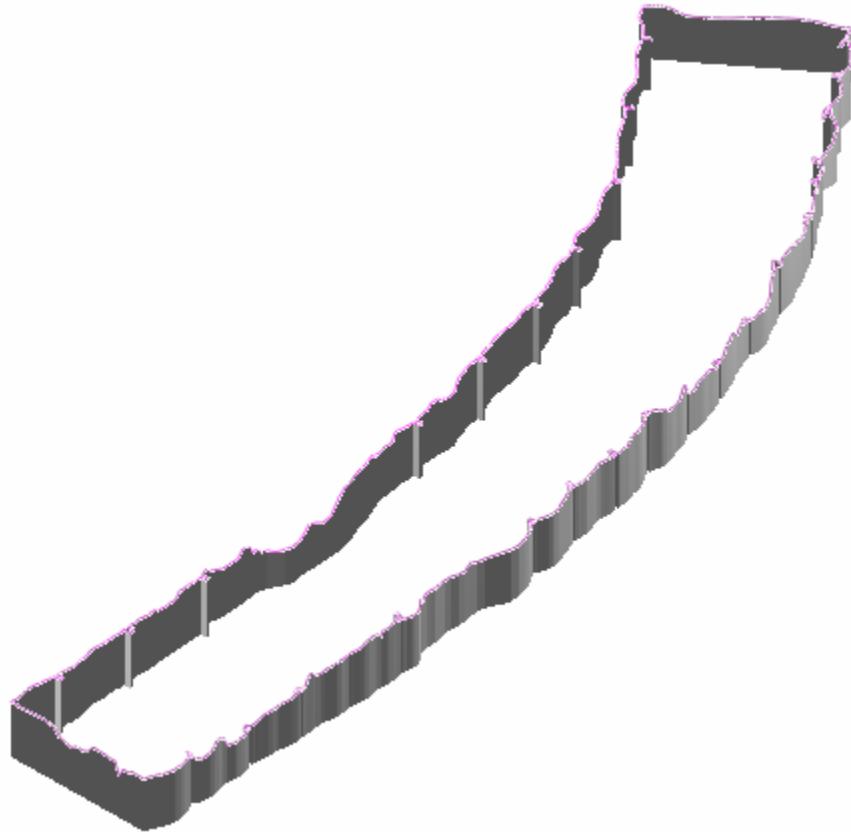


Figure 4-103

4.8.3 Shape Files

After loading the shape files, the users can set the display parameters and view the attributes table.

To set the display parameters,

- First select a shape file on the left **Project** view.

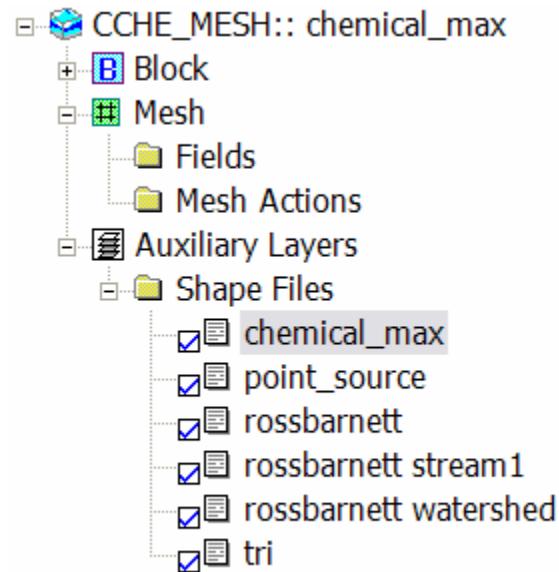


Figure 4-104

- Then Right-click the selected shape file. In the popup menu, select “Settings...” You can set the point size, point color, line width, line color and filled color. You can also choose to display the current selected shape file on the top layer.

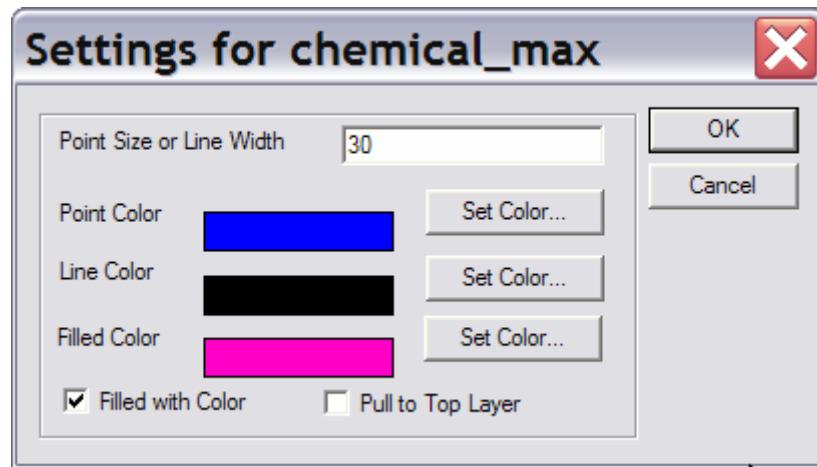


Figure 4-105

To view the attributes table,

- First select a shape file on the left **Project** view.
- Then Right-click the selected shape file. In the popup menu, select “**Attributes...**”

Attributes Table: chemical_max

Record #	Chemicals	to_water	to_air	to_earth
1	1,1,1-TRICHLOROETHANE;	0	25542	0
2	AMMONIA	0	0	220
3	AMMONIA	0	114903	0
4	AMMONIA	1600	0	0
5	CHROMIUM	0	0	349
6	CHROMIUM	0	1410	0
7	COPPER	0	0	362
8	COPPER	11	0	0
9	COPPER	0	110	0
10	DIOXIN AND DIOXIN-LIKE COMPOUNDS	0	0	0

Selected Record Plot Attributes

Selected Record: 1 Export... X Values: Add Series Plot >>

Highlight selected record Y Values: Delete All

Figure 4-106

- In **Attributes Table**, the users can highlight a selected record which corresponds to a shape object, and the users can also plot attributes.
 - To plot attributes, first you need to select the “**X values**” which lists all the available attributes, then select the “**Y Values**”, and then select “**Add Series**”; to plot multiple attributes, you can repeat the above procedure. Finally, click “**Plot >>**”, and the attributes will be displayed with curves. The caption of this button will be changed into “**Close Plot <<**” and the window will be expanded to show the XY plot..
 - Select “**Delete All**” to delete all curves, and then you can restart to plot some new attributes.

- Select “Close Plot <<” to close the XY Plot and restore the original size of the window.

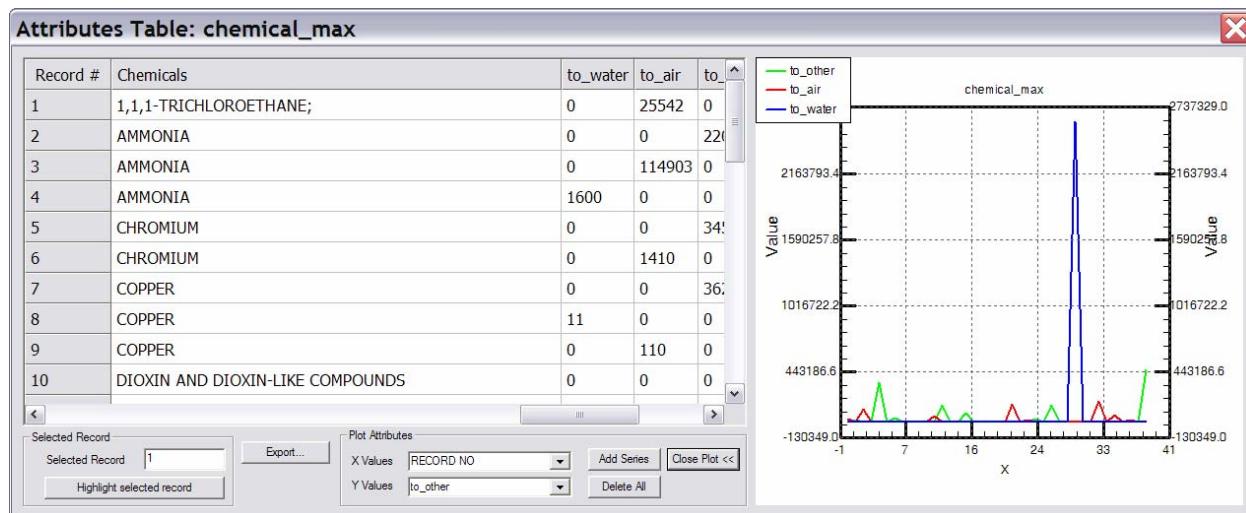


Figure 4-107

4.8.4 Display Options

The CCHE-MESH allows you to customize your own drawing by setting the display options. To set the display options, select Options... in View menu or on View Tool toolbar.

As shown in Figure 4-97, there are five pages in the **Display Options** window.

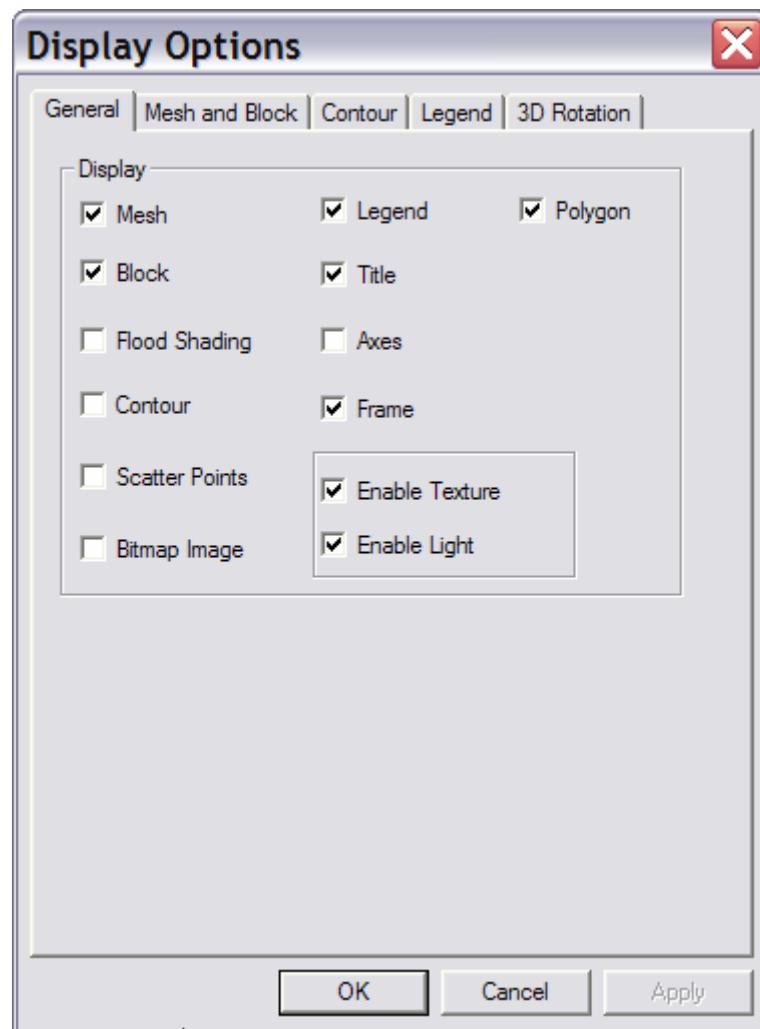


Figure 4-108

On the first page, you can show/hide different plotting by checking or unchecking the corresponding item, such as Mesh, Block, etc. You can also enable or disable the **Texture** and **Light** effects.

On the **Mesh and Block** page, you can set the properties for the mesh and block, such as **Line Width**, **Line Color**, **Highlight Color**, etc.

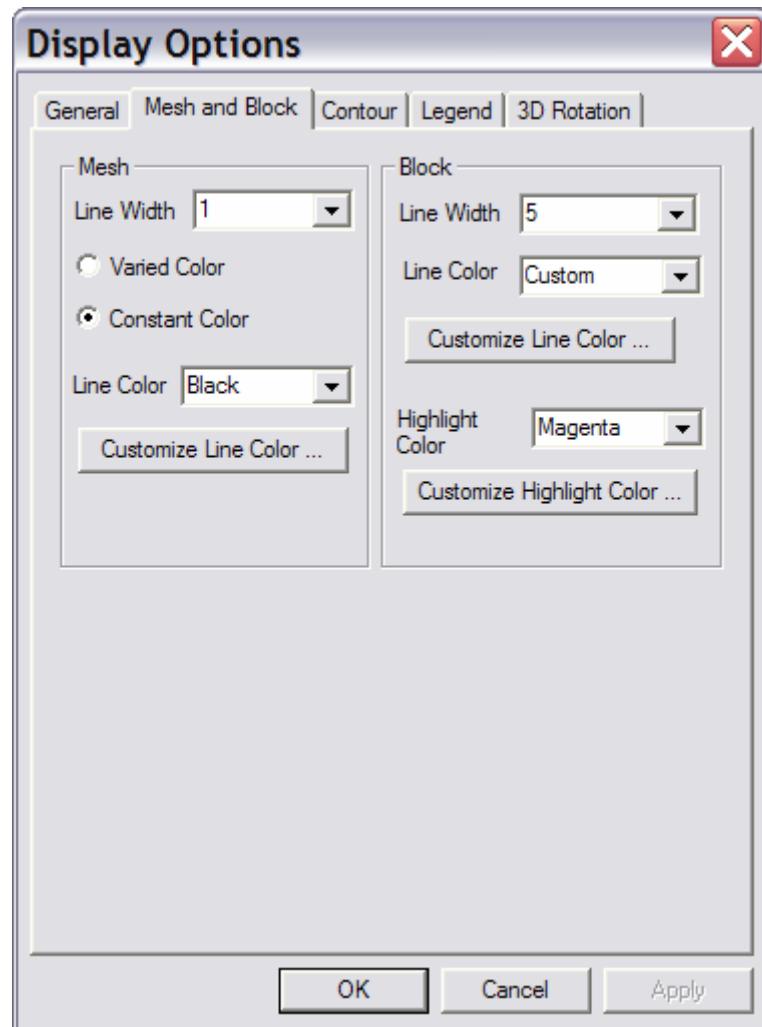


Figure 4-109

On the **Contour** page, you can set the properties for the contour. These properties are divided into three groups.

- **Contour Levels:** In this group, you can set the current **Minimum** and **Maximum** values for the current selected variable; and, you can also set the contour type: plotting by the number of levels or by the interval.

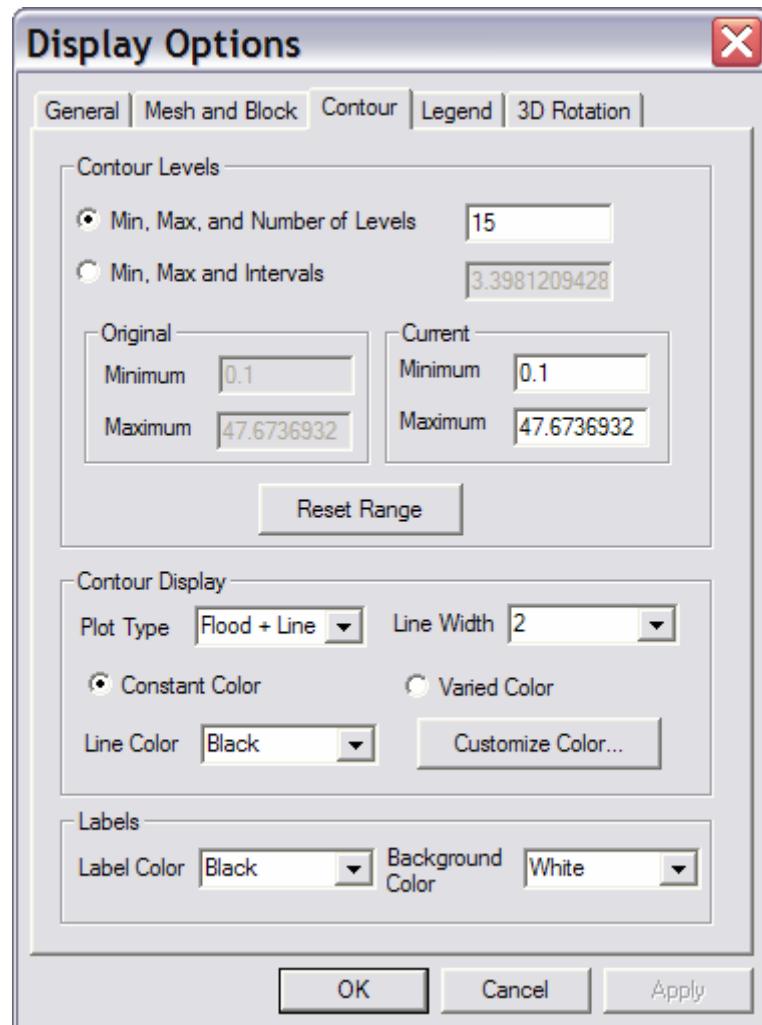


Figure 4-110

- **Contour Display:** This group allows you to set the display properties, such as **Plot Type**, **Line Width**, **Line Color**, etc.
- **Labels:** In this group, you can set the **Label Color** and the **Background Color** for the contour labels.

On the **Legend** page, you can set the properties for the legend including the **Orientation**, the **Color Type**, the **Width** and **Height**, and the digit **Precision**.

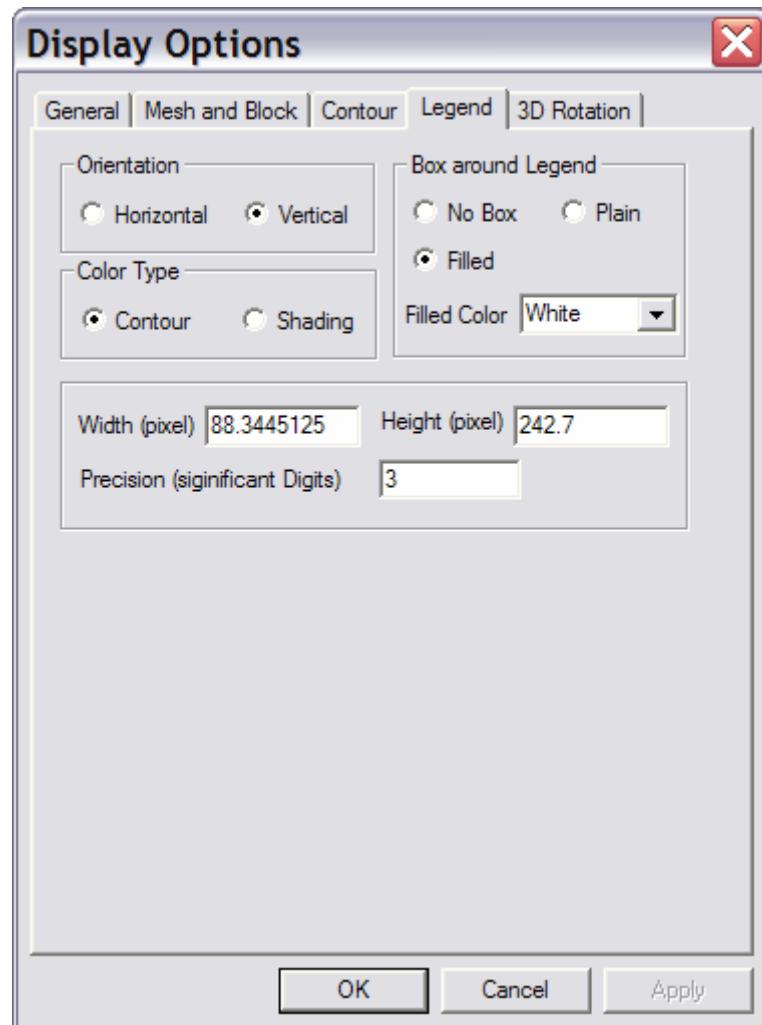


Figure 4-111

On the last page, the **3D Rotation** properties can be set. Only if the **3D view** is active will this page enabled. You can set the rotation angles in x, y, z direction. You can also view the specific planes, such as **XY**, **-XY**, **YZ**, **-YZ**, **XZ**, and **-XZ** planes.

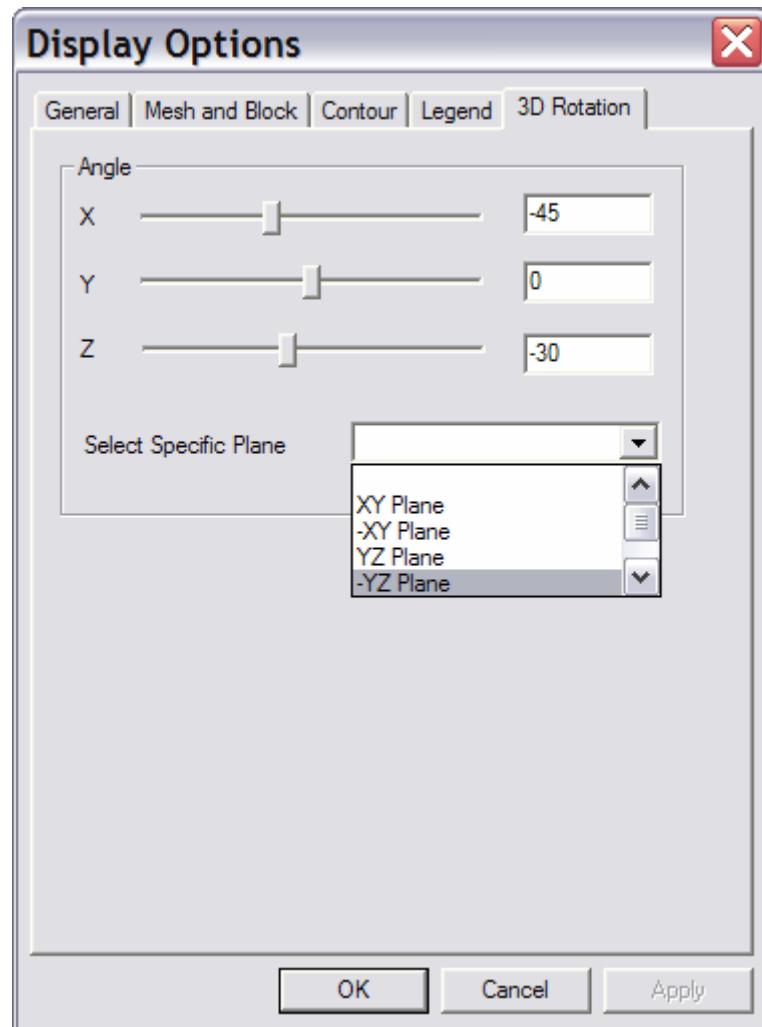


Figure 4-112

4.8.5 Background

The CCHE-MESH provides the functions to import and edit the background image.

To import a background image, select **Bitmap Image (*.bmp)** from **Import** submenu on **File** menu.

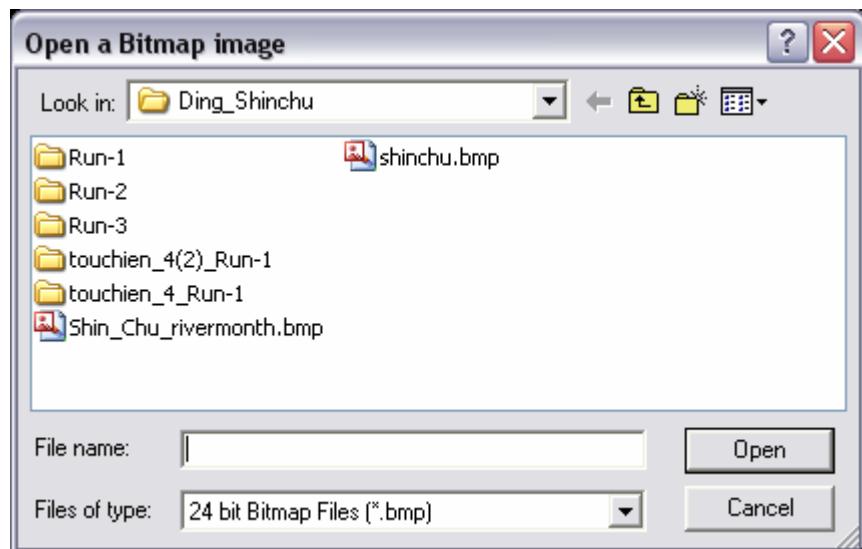


Figure 4-113

After you import a bitmap image successfully, you can use the following toolbar to edit it.



Figure 4-114

To do the coordinates transformation, you need to define two reference points on the image.

- First select **R+** to enter the editing status.
- Define the first reference point.

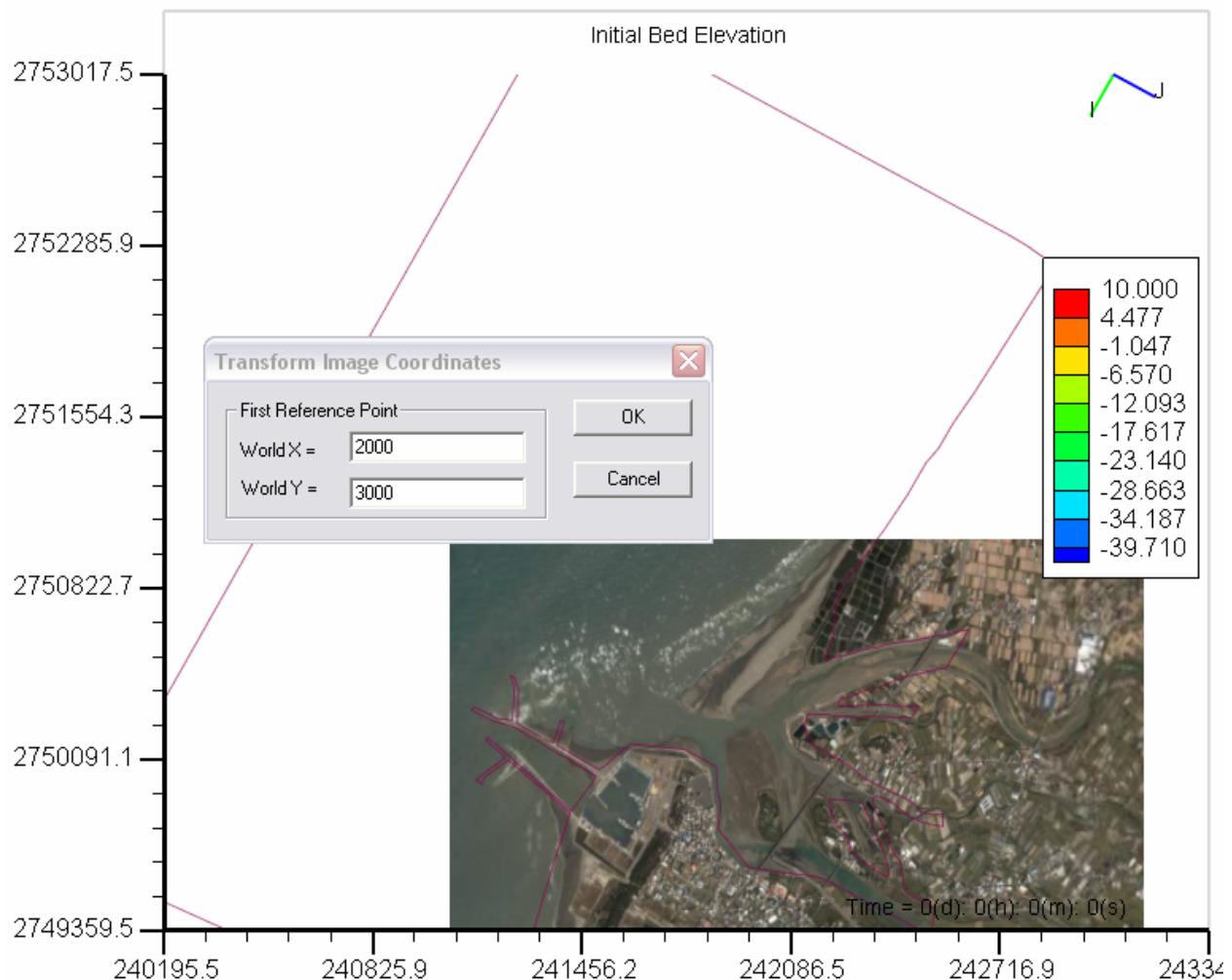


Figure 4-115

- Define the second reference point.

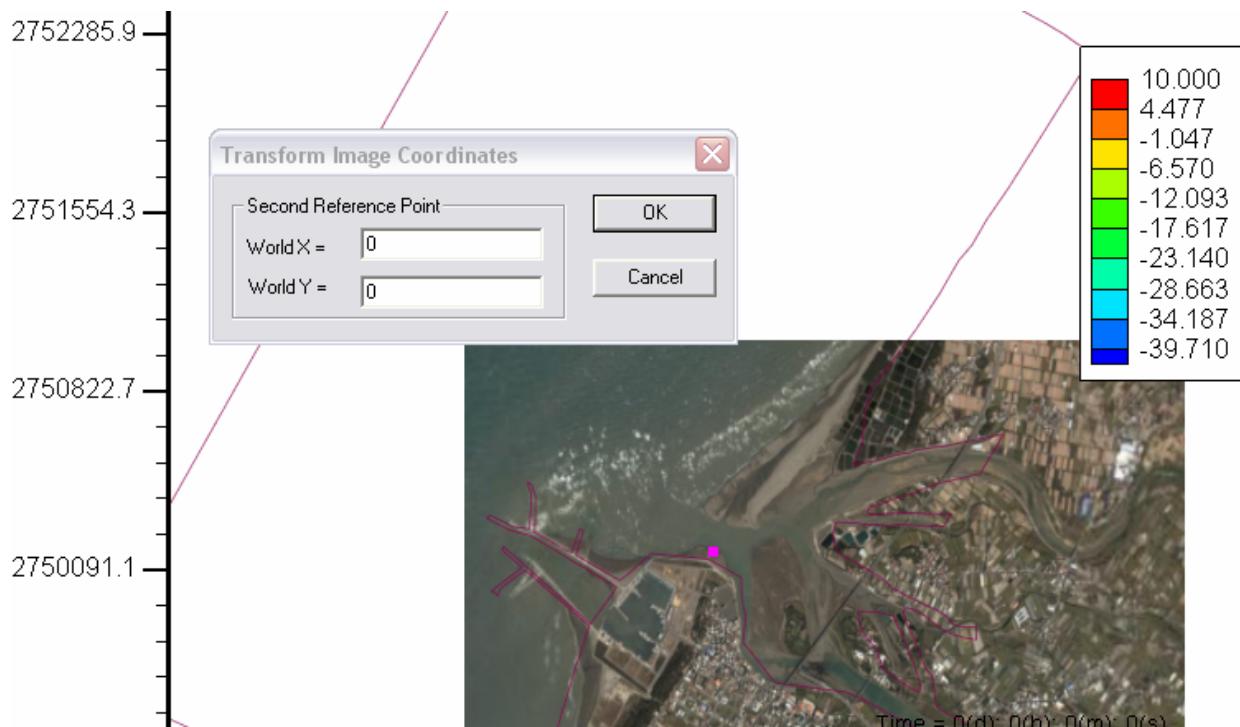


Figure 4-116

- **Save Transformation Information:** Select save the transformation information into 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

4.9 File Formats

There are five files used in the CCHE-MESH: the mesh block boundary file (*.mesh_mb), the topography database file (*.mesh_xyz), the mesh workspace file (*.mesh_wsp), the measured cross section file (*.mesh_mcs), and the output mesh file for the CCHE2D and CCHE3D models (*.geo).

You need to be familiar with the formats of these files except the mesh workspace file. The formats of these files in Fortran are reviewed in detail as follows for convenient reference.

4.9.1 Topography Database File (*.mesh_xyz)

The geometry database file is used to interpolate bed elevation of mesh nodes. The format of this file is very simple and the reading format in Fortran code is as follows.

```

Open(12, file = "test.mesh_xyz")
Read(12,*)nip           //---number of points
DO k = 1, nip
    Read(12, *)xip(k), yip(k), zip(k) //---(x,y,z) of each point
ENDDO

```

4.9.2 Measured Cross Sections File (*.mesh_mcs)

The measured cross section data file is used to refine the original geometry database. The format in Fortran code is as follows.

```

Open(13, file = "test.mesh_mcs")
Read(13,*) ncs           //---number of cross sections
DO k = 1, ncs
    Read(13,*) lcs, np(k), flag
    //---label and number of points of cross section k, the flag defining the direction is optional,
    //you may define the direction from left bank to right bank as 0, and the inverse direction is 1,
    //or vice versa.
    DO j = 1, np(k)
        //---(x,y,z) of each points
        Read(13,*) xcs(k,j), ycs(k,j), zcs(k,j)
    ENDDO
ENDDO

```

4.9.3 Mesh Geo File (*.geo)

The mesh geo file is an output file of CCHE mesh generator. The format in Fortran code is as follows.

```
Open(14, file = "test.geo")
Write(14,*)jmax, imax //---maximum number of i lines and j lines
DO i = 1, imax
    DO j = 1, jmax
        Write(14,*)xp(i ,j), yp(i ,j), zs(i ,j), zp(i ,j), id(i ,j), rough(i ,j)
    ENDDO
ENDDO
```

References

- Thompson, J.F., Thames, F.C., and Mastin, C.W.(1977), “TOMCAT—A code for numerical generation of boundary-fitted curvilinear coordinate system on fields containing any number of arbitrary two-dimensional bodies,” *J. Comput. Phys.* 1977; 24:274-302.
- Brackbill, J.U. and Saltzman, J.S.(1982), “Adaptive zoning for singular problems in two-dimensions,” *J. Comp. Phys.*, 1982; 46:342.
- Ryskin, G., and Leal, L.G (1983), “Orthogonal mapping”, *J.Comput. Phys.* 1983; 50(1):71-100.
- Zhang, Yaxin and Jia, Yafei (2005). “CCHE-MESH Mesh Generator---Technical Report and Users Manual version 2.6”, National Ceneter for Computational Hydroscience and Engineering, Technocal Report No.NCCHE-TR-2005-05, September, 2005.
- Zhang, Yaxin, Jia, Yafei, and Wang, S.Y.Sam (2007). “Two-dimensional Adaptive Mesh Generation.” *Int'l Journal for Numerical Methods in Fluids*, 2007; 54(11): 1327-1350.
- Zhang, Yaxin, Jia, Yafei, and Wang, S.Y.Sam (2006b), “2D Mesh Generation with Controls Distortion Functions.” *Journal of Computational Physics*, 2006; 218(2): 549-571.
- Zhang, Yaxin, Jia, Yafei, and Wang, S.Y.Sam (2006a) “Structured Mesh Generation with Smoothness Controls.” *Int'l Journal for Numerical Methods in Fluids*, 2006; 51: 1255-1276.
- Zhang, Yaxin, Jia, Yafei, and Wang, S.Y.Sam (2004) “2D Near-orthogonal Mesh Generation.” *Int'l Journal for Numerical Methods in Fluids*, 46(9): pp. 685-707, 2004
- Zhang, Yaxin. (2004). “Development of Mesh Generation and Multi-block Algorithm for Numerical Models”, *Ph.D Dissertation*, National Center for Computational Hydroscience and Engineering, The University of Mississippi, May, 2004.