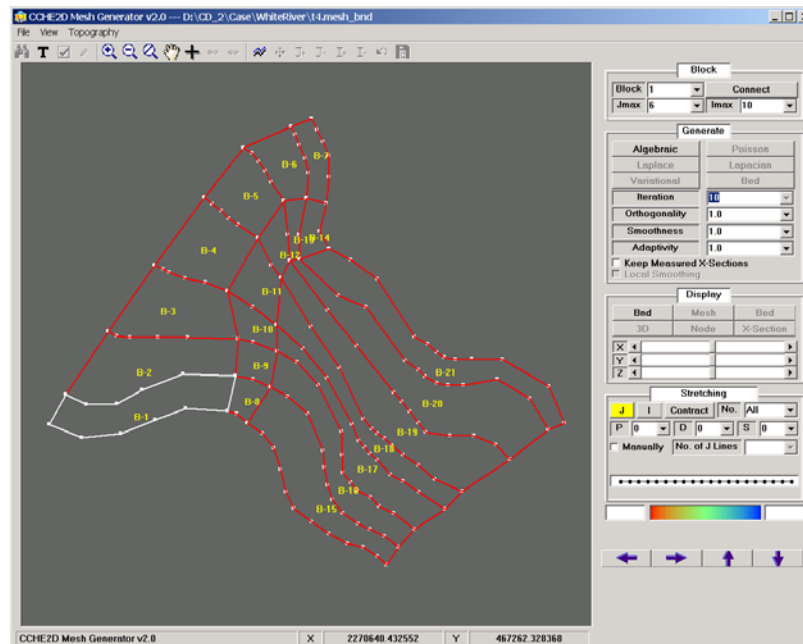


National Center for Computational Hydroscience and Engineering

CCHE2D Mesh Generator and User's Manual (v 2.00)

Technical Report No. NCCHE-TR-2002-5
Yaoxin Zhang and Yafei Jia

November, 2002



School of Engineering
The University of Mississippi
University, MS 38677

CCHE2D Mesh Generator and User's Manual

Version 2.0

NCCHE-TR-2002-5

Project Director and Principle Investigator

Sam S.Y. Wang, Ph.D.

F.A.P. Barnard Distinguished Professor and Director
Center for Computational Hydroscience of Engineering
The University of Mississippi
University, MS 38677

Senior Investigator

Yafei Jia , Ph.D.

Research Associate Professor
Center for Computational Hydroscience of Engineering
The University of Mississippi
University, MS 38677

November 2002

Abstract

This documentation describes the methodology and the Graphic Users' Interface of a mesh generator, CCHE2D Mesh Generator.

Mesh generation for computational hydrodynamic problems is usually a complicated and time-consuming task which often discretise the irregular geometry of river channels, reservoirs to a system of finite sized elements on which the numerical simulations are carried out. Because of this, one has to combine and utilize a collection of methods for different mesh generation needs, and these methods and their tedious manipulation has to be handled with a GUI to have high efficiency. Based on these considerations, CCHE2D Mesh Generator has been developed. The current version not only has an interactive interface but also combines several methods for users to select.

Since the current NCCHE models use structured mesh, this version of CCHE2D Mesh Generator only produces the structured mesh. Both the sophisticated methods of solving partial differential equations and simple algebraic method are available. The later is based on a flexible and powerful two-direction stretching function and the former are a Poisson equation based solver whose source terms are based on control functions and the variational approach with several optimal objective controls. To take advantage of the GUI, a mesh can be generated by matching small mesh pieces together.

The GUI is newly developed for the CCHE2D Mesh Generator, which is in the process of testing and further improving. Users are encouraged to contact NCCHE if difficulties are encountered in generating a mesh.

Contents

Title	Page No.
Chapter 1	1
Introduction	1
Chapter 2	2
Mesh Generation	2
<i>2.1 Algebraic Mesh</i>	<i>2</i>
<i>2.1.1 Stretching Function</i>	<i>3</i>
<i>2.1.2 Generation Steps</i>	<i>5</i>
<i>2.2 Poisson Equation</i>	<i>7</i>
<i>2.3 Variational Formulation</i>	<i>9</i>
<i>2.4 Smoothing Methods</i>	<i>11</i>
<i>2.5 Interpolation of Bed Elevation</i>	<i>14</i>
Chapter 3	16
Interface of CCHE2D Mesh Generator	16
<i>3.1 Introduction of Interface</i>	<i>16</i>
<i>3.1.1 Groups of Main Window</i>	<i>17</i>
<i>3.1.2 Control Panel</i>	<i>25</i>
<i>3.1.3 Menus</i>	<i>26</i>
<i>3.2 How to Generate a Mesh</i>	<i>28</i>
<i>3.2.1 Generate Mesh from Topographic Database file.....</i>	<i>28</i>
<i>3.2.2 Generate Mesh from Topographic Image.....</i>	<i>35</i>
<i>3.2.3 Generate Mesh from Boundary File</i>	<i>38</i>
<i>3.2.4 Generate Algebraic Mesh</i>	<i>40</i>
<i>3.2.5 Generate Near-Orthogonal Adaptive Mesh</i>	<i>50</i>
<i>3.2.6 Generate Variational Adaptive Mesh</i>	<i>51</i>

3.2.7 <i>Generate Smooth Mesh</i>	51
3.3 <i>How to Use Stretching Function</i>	51
3.3.1 <i>Apply Stretching Function in J Direction</i>	52
3.3.2 <i>Apply Stretching Function in I Direction</i>	54
3.4 <i>How to Edit Mesh</i>	55
3.4.1 <i>Edit Boundary</i>	55
3.4.2 <i>Adjust Mesh Node's or Line's Location</i>	56
3.4.3 <i>Modify the Properties of Mesh Node</i>	57
3.5 <i>View Mesh In Different Ways</i>	58
3.5.1 <i>View Bed Topography</i>	58
3.5.2 <i>View Bed Topography with Mesh on</i>	59
3.5.3 <i>View 3D Mesh</i>	60
3.5.4 <i>View Mesh Nodes</i>	61
3.5.5 <i>View Cross Sections</i>	62
3.5.6 <i>Add Background</i>	64
3.6 <i>System Requirements</i>	66
3.7 <i>Review of Formats of Files</i>	66
3.7.1 <i>The Mesh Boundary File (*.mesh_bnd)</i>	66
3.7.2 <i>The bathymetry Database File (*.mesh_xyz)</i>	66
3.7.3 <i>The Measured Cross Section File (*.mesh_mcs)</i>	69
3.7.4 <i>The Mesh Geo File (*.geo)</i>	70
 Conclusions	 71
References	

Chapter 1

Introduction

The mathematical modeling of hydrodynamics and sediment and pollutant transport involves a set of partial differential equations (Navier-Stokes' Equations) for mass, momentum, pollutant and energy conservation. Since they are highly non-linear, the analytical solutions in general cannot be found. Numerical solutions can be computed based on discretized equations, and the discretization is dependent on the mesh which approximates the entire irregular complicated domain.

Since mesh generation is a time-consuming and tedious process which requires handling large amounts of data, transformation, and solving partial differential equations, a mesh generation is necessary which can not only make all the involved calculations, but also handles the parameters input and the results displayed. More importantly, mesh generation is a process that is problem-dependent and needs human intelligence involvement. It is therefore an interactive process. The GUI of the mesh generator provides such a tool for researchers and engineers to generate the mesh efficiently.

Both the mathematical methods involved and the GUI of NCCHE2D Mesh Generator are introduced. Chapter 2 discusses in detail the mathematical methods of mesh generation, including the algebraic method, the Poisson-equation-based numerical method, the Variational numerical method and the Laplacian smoothing methods, etc. Chapter 3 introduces the GUI. A typical example is given to demonstrate how to generate a mesh step by step.

This mesh generator has been developed under the supervision of Dr. Sam S.Y. Wang, Director of NCCHE, and Dr. Yafei Jia, Senior Investigator of NCCHE. Mr. Yaixin Zhang, a Ph.D. student of NCCHE, is in charge of implementation.

Chapter 2

Mesh Generation

There are two main types of meshes used in numerical calculation: structured and 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.

There are two kinds of methods to generate structured meshes: numerical methods and algebraic methods. The numerical methods solve the partial differential equations, usually Poisson equations, to generate boundary-fitted mesh, while the algebraic methods, determine the mesh directly according to some desired nodal distribution. The numerical methods are global approaches, which can provide very good meshes with smooth transition of nodal distribution, smoothness, and orthogonality guaranteed. Although the algebraic method can always be used to resolve a mesh, its' global smoothness, even local smoothness, uniformity properties are not guaranteed. As a result, smoothing methods are often applied to optimize the mesh. For cases with complicated domain geometry, the multi-block approach could be applied to simplify a complex mesh to several small meshes with simpler geometry.

In practice, the algebraic method, numerical method and multi-block method are used in combination to take the advantage of each other. The interactive GUI is important to make the combination efficiently. In this chapter, the mesh generation methodologies are described and the GUI will be introduced in the next chapter.

2.1 Algebraic Mesh

The algebraic generator adopted here is to create the mesh points in the computational domain by interpolating the nodes according to the nodal distribution along the boundaries of the domain. Stretching functions are used to distribute computational nodes along boundaries with several control parameters. The distribution for a specific problem should be determined by experienced engineers according to the conditions of flow, boundary shape, etc. Interpolation is simply to connect the corresponding nodes along opposite sides with straight lines.

2.1.1 Stretching Function

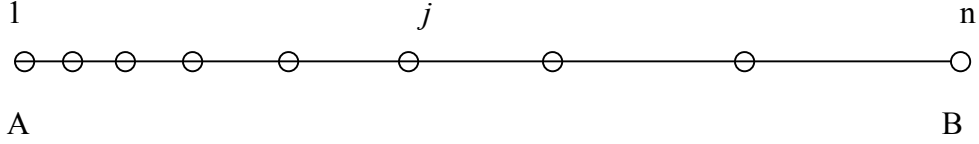
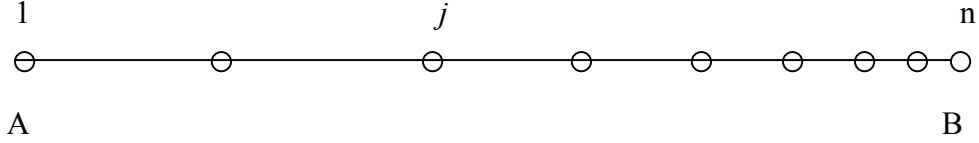
The stretching function used in the algebraic method is to control the distribution of points on one curved or straight line.

Equation (2-1) shows the so-called one-direction stretching function. This kind of stretching function can make the distribution of points along on one line with monotonically increasing or decreasing spacing.

$$S_j = \frac{k^{j-1} - 1}{k^{n-1} - 1} \quad (\text{for } k \neq 1) \quad (2-1a)$$

$$S_j = \frac{j-1}{n-1} \quad (\text{for } k = 1) \quad (2-1b)$$

where S_j is the relative location and k is the density parameter. When $k < , =$, or >1 , the spacing of points will be decreased, unchanged, and increased respectively (Fig-2-1).

Fig 2-1(a) $k > 1$ (one-direction stretching function)Fig 2-1(b) $k < 1$ (one-direction stretching function)

In this version of CCHE mesh generator, an improved more flexible and powerful two-direction stretching function is used.

$$S_j = \sum_{i=1}^{j-1} \left(\frac{2}{e^{\eta} + e^{-\eta}} \right)^P / \sum_{i=1}^{n-1} \left(\frac{2}{e^{\eta} + e^{-\eta}} \right)^P \quad (2-2a)$$

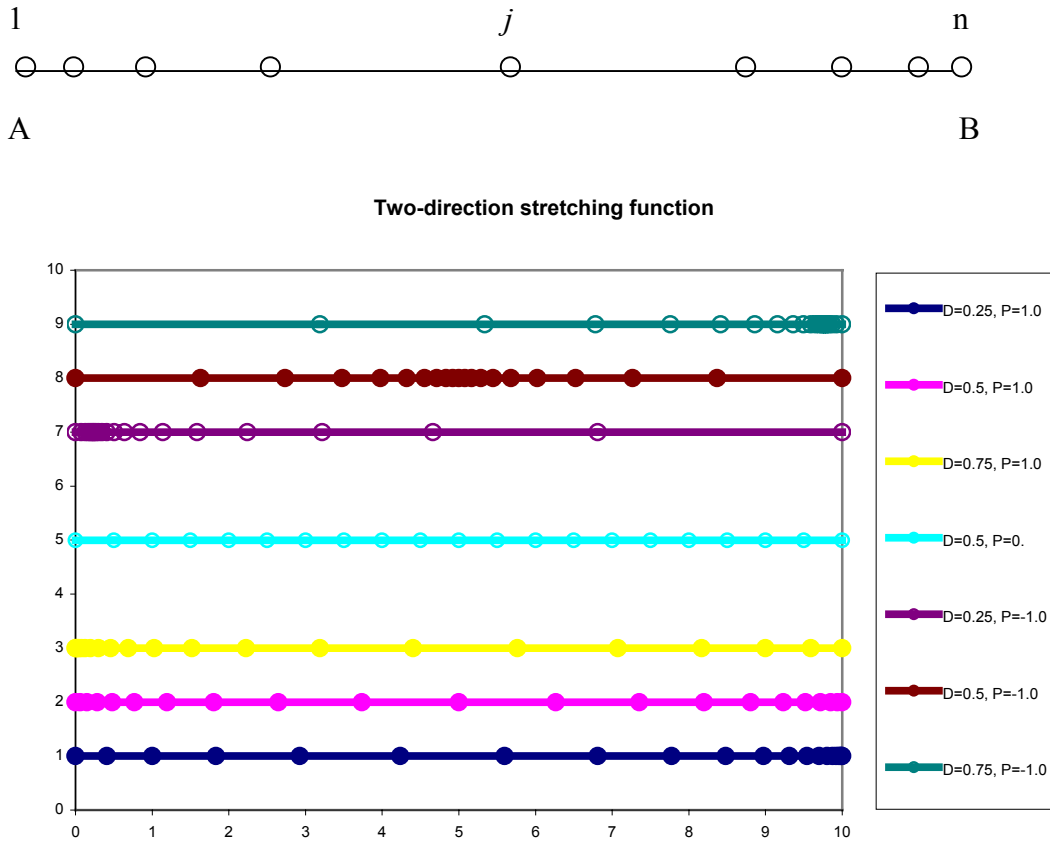
$$\eta = \left[\frac{j-1}{n} - D \right] \times S \quad (2-2b)$$

where S_j is the relative location; j is the label of nodal points; n is the total number of points; P ($= -1, 0, 1$) is the exponential coefficient, called **Power parameter**. It determines if the points will be contracted or repulsed or uniformly distributed; D is the deviational coefficient, $0 < D < 1$, called **Deviation parameter**. It determines the relative location where the points will be contracted or repulsed; S is the parameter used to control stretching degree, called **Scale parameter**. It determines in what degree the points will be contracted or repulsed.

The effects of **P** and **D** are shown in the Fig 2-2.

In the following figure, the location of any node along this line AB is calculated by

$$x_j = x_A + (x_B - x_A) \cdot S_j \quad (2-3)$$

Fig 2-2 The Effects of P and D

2.1.2 Procedures to Generate Algebraic Mesh

It has been mentioned that a mesh for a computational domain can be generated as a whole or as a collection of several separated sub-domains (blocks). The generation steps illustrated here applies to one sub-domain (single block) if several sub-domains (multi-blocks) are involved, or to the whole domain if it is not divided. Basically, four steps are needed to generate an algebraic mesh.

Step 1: Place the geometry control points along the boundaries. Then the domain is divided into several subsections (Fig 2-3). In the CCHE mesh generator, two kinds of boundaries are identified: external boundary and internal boundary. According to the

location, external boundary consists of four kinds of boundaries, namely, left boundary, right boundary, top boundary, and bottom boundary (Fig 2-4).

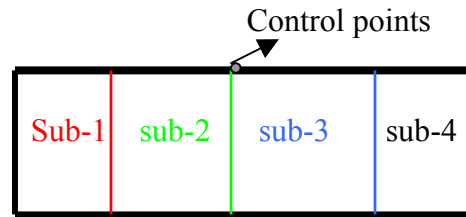


Fig 2-3 Subsections

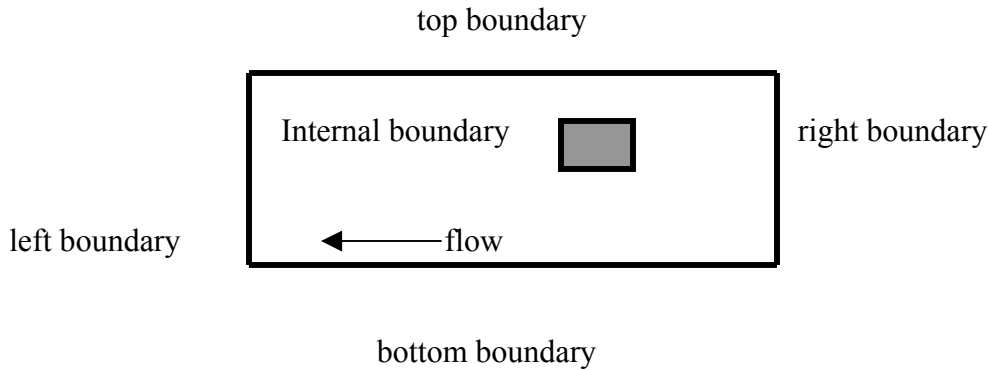


Fig 2-4 Boundaries

Step-2: Distribute the mesh control lines along the top and bottom boundaries (Fig 2-5). The stretching function is used to distribute the mesh control lines along the boundaries.

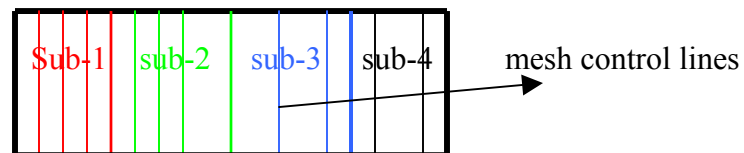


Fig 2-5 Mesh Control Lines

Step-3: Interpolate the interior nodes (Fig 2-6). The stretching function is used to distribute the interior nodes in the transverse direction.

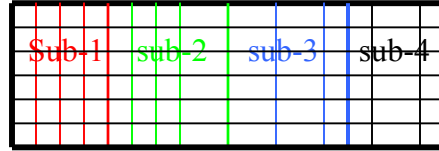


Fig 2-6 Interior nodes

Step-4: Interpolate the bed elevation (discussed in detail later).

2.2 Modified Poisson Equation

The partial differential equation commonly used for the numerical mesh generation is the Poisson equation in the form of

$$\frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} = P(\xi, \eta), \quad \frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = Q(\xi, \eta) \quad (2-4)$$

where (ξ, η) is computational coordinated system and (x, y) is physical coordinated system; P, Q are control functions for the clustering of points.

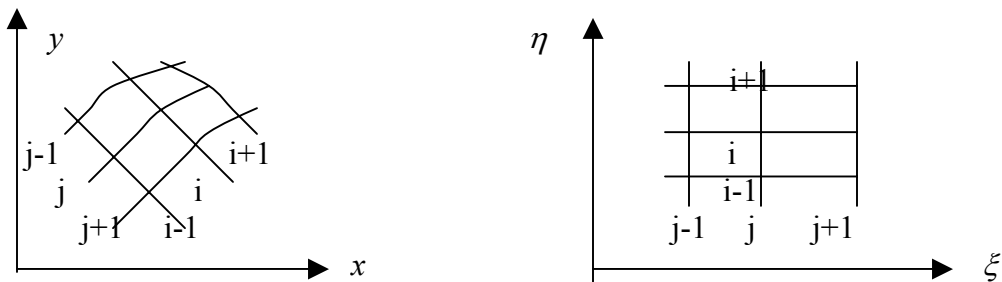


Fig 2-7 Physical domain and computational domain

Transforming the above equation (2-4), we obtain

$$\begin{aligned}
\alpha \frac{\partial^2 x}{\partial \xi^2} - \beta \frac{\partial^2 x}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 x}{\partial \eta^2} + \delta \left(P \frac{\partial x}{\partial \xi} + Q \frac{\partial x}{\partial \eta} \right) &= 0 \\
\alpha \frac{\partial^2 y}{\partial \xi^2} - \beta \frac{\partial^2 y}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 y}{\partial \eta^2} + \delta \left(P \frac{\partial y}{\partial \xi} + Q \frac{\partial y}{\partial \eta} \right) &= 0
\end{aligned} \tag{2-5a}$$

where $\alpha = g_{22}, \beta = g_{12}, \gamma = g_{11}, \delta = g$.

$$g = \begin{vmatrix} (x_\xi^2 + y_\xi^2) & (x_\xi x_\eta + y_\xi y_\eta) \\ (x_\xi x_\eta + y_\xi y_\eta) & (x_\eta^2 + y_\eta^2) \end{vmatrix} \tag{2-5b}$$

$$\text{The orthogonal condition is: } \beta = g_{12} = 0 \tag{2-6}$$

Define P and Q in the following formulae,

$$P = \frac{1}{h_1 h_2} f_\xi, \quad Q = \frac{1}{h_1 h_2} \left(\frac{1}{f} \right)_\eta \tag{2-7a}$$

$$\text{where } h_1 = g_{11}^{1/2}, \quad h_2 = g_{22}^{1/2} \tag{2-7b}$$

$$f = \frac{h_2}{h_1} = \left(\frac{x_\eta^2 + y_\eta^2}{x_\xi^2 + y_\xi^2} \right)^{1/2} \tag{2-7c}$$

$$f_\xi = \frac{\left(\frac{x_\xi^2 + y_\xi^2}{x_\eta^2 + y_\eta^2} \right)^{1/2} (x_\eta x_{\eta\xi} + y_\eta y_{\eta\xi}) - \left(\frac{x_\eta^2 + y_\eta^2}{x_\xi^2 + y_\xi^2} \right)^{1/2} (x_\xi x_{\xi\xi} + y_\xi y_{\xi\xi})}{x_\xi^2 + y_\xi^2} \tag{2-7d}$$

$$\left(\frac{1}{f} \right)_\eta = \frac{\left(\frac{x_\eta^2 + y_\eta^2}{x_\xi^2 + y_\xi^2} \right)^{1/2} (x_\xi x_{\eta\xi} + y_\xi y_{\eta\xi}) - \left(\frac{x_\xi^2 + y_\xi^2}{x_\eta^2 + y_\eta^2} \right)^{1/2} (x_\eta x_{\eta\eta} + y_\eta y_{\eta\eta})}{x_\eta^2 + y_\eta^2} \tag{2-7e}$$

Using centered difference scheme to discretize the equation (2-5a), we obtain:

$$\alpha'(x_{j-1,i} - 2x_{j,i} + x_{j+1,i}) + \gamma'(x_{j,i-1} - 2x_{j,i} + x_{j,i+1}) + 0.5\delta'P(x_{j+1,i} - x_{j-1,i}) + 0.5\delta'Q(x_{j,i+1} - x_{j,i-1}) = 0 \quad (2-8a)$$

$$\alpha'(y_{j-1,i} - 2y_{j,i} + y_{j+1,i}) + \gamma'(y_{j,i-1} - 2y_{j,i} + y_{j,i+1}) + 0.5\delta'P(y_{j+1,i} - y_{j-1,i}) + 0.5\delta'Q(y_{j,i+1} - y_{j,i-1}) = 0 \quad (2-8b)$$

where

$$\alpha' = 0.25[(x_{j,i+1} - x_{j,i-1})^2 + (y_{j,i+1} - y_{j,i-1})^2] \quad (2-8c)$$

$$\gamma' = 0.25[(x_{j+1,i} - x_{j-1,i})^2 + (y_{j+1,i} - y_{j-1,i})^2] \quad (2-8b)$$

$$\delta' = [(x_{j+1,i} - x_{j-1,i})(y_{j,i+1} - y_{j,i-1}) - (x_{j,i+1} - x_{j,i-1})(y_{j+1,i} - y_{j-1,i})]^2 / 16 \quad (2-8d)$$

Rearranging the equation (2-8), we obtain:

$$x_{j,i}^n = \frac{\alpha'}{2\alpha' + 2\gamma'}(x_{j-1,i} + x_{j+1,i}) + \frac{\gamma'}{2\alpha' + 2\gamma'}(x_{j,i-1} + x_{j,i+1}) + \frac{0.5\delta'}{2\alpha' + 2\gamma'}P(x_{j+1,i} - x_{j-1,i}) + \frac{0.5\delta'}{2\alpha' + 2\gamma'}Q(x_{j,i+1} - x_{j,i-1}) \quad (2-9a)$$

$$y_{j,i}^n = \frac{\alpha'}{2\alpha' + 2\gamma'}(y_{j-1,i} + y_{j+1,i}) + \frac{\gamma'}{2\alpha' + 2\gamma'}(y_{j,i-1} + y_{j,i+1}) + \frac{0.5\delta'}{2\alpha' + 2\gamma'}P(y_{j+1,i} - y_{j-1,i}) + \frac{0.5\delta'}{2\alpha' + 2\gamma'}Q(y_{j,i+1} - y_{j,i-1}) \quad (2-9b)$$

In order to generate 2D near-orthogonal mesh with high smoothness and adaptivity, we modified the conventional Poisson equation by introducing effect control factor and contribution factor. The effect control factor combines the conformal mapping and the orthogonal mapping together, which is capable of preventing mesh distortion and overlapping effectively and also enhancing the convergence of mesh computation due to the fact that Laplace equation is numerically more stable than Poisson equation. With contribution factor, adaptive mesh generation, i.e., depth-adaptive mesh, temperature-adaptive mesh, etc., becomes possible.

The modified Poisson Equation reads:

$$x_{i,j}^n = \frac{\alpha'}{T}(c_{i-1,j} \cdot x_{i-1,j} + c_{i+1,j} \cdot x_{i+1,j}) + \frac{\gamma'}{T}(c_{i,j-1} \cdot x_{i,j-1} + c_{i,j+1} \cdot x_{i,j+1}) + r \cdot \frac{0.5\delta'}{T}P(x_{i+1,j} - x_{i-1,j}) + r \cdot \frac{0.5\delta'}{T}Q(x_{i,j+1} - x_{i,j-1}) \quad (2-10a)$$

$$y_{i,j}^n = \frac{\alpha'}{T}(c_{i-1,j} \cdot y_{i-1,j} + c_{i+1,j} \cdot y_{i+1,j}) + \frac{\gamma'}{T}(c_{i,j-1} \cdot y_{i,j-1} + c_{i,j+1} \cdot y_{i,j+1}) + r \cdot \frac{0.5\delta'}{T}P(y_{i+1,j} - y_{i-1,j}) + r \cdot \frac{0.5\delta'}{T}Q(y_{i,j+1} - y_{i,j-1}) \quad (2-10b)$$

$$T = \alpha'(c_{i-1,j} + c_{i+1,j}) + \gamma'(c_{i,j-1} + c_{i,j+1}) \quad (2-10c)$$

where r is the effect control factor and $0 \leq r \leq 1$; and $c_{i,j}$ is the contribution factor. Note that Equation (2-10) is the same as (2-9) when $r = 1$ and $c_{i,j} = 1$.

When generating adaptive mesh, the contribution factor is evaluated by the following equation:

$$c_{i,j} = 1 + \left(1 - \frac{z_{i,j} - z_{\min}}{z_{\max} - z_{\min}}\right) \cdot r_a \quad (2-11)$$

where r_a is the adaptivity parameter and $r_a \geq 0$; and $z_{i,j}$ is bed elevation.

2.3 Variational Laplace Equation

The variational Laplace equation is adopted in this version of CCHE2D Mesh Generator. Three indices could be used to evaluate the quality of a mesh system: uniformity, orthogonality, and adaptivity. Uniformity indicates the mesh spacing is uniform distributed; Orthogonality is a measure of how much the mesh lines are perpendicular to each other; and adaptivity is the nodal distribution to areas where higher resolution and accuracy are desired.

The *adaptivity* of the mesh is measured by the functional:

$$I_w = \int_D w(x, y) J dA \quad (2-11)$$

where $J = x_\xi y_\eta - x_\eta y_\xi$ (2-12)

is the Jacobian, which represents the area of a computational cell in 2-dimensions.

When this integral is minimized, $w(x, y)J$ (with $w(x, y) > 0$) should have a uniform distribution, so that where the weighting function is large the mesh size J should be small.

The weighting function w can be selected, which is usually evaluated by water depth and bed slope. If the numerical solutions such as concentration or velocity gradients are selected, the mesh shall be changed dynamically with the numerical solution.

Uniformity of the mesh is measured by the functional

$$I_s = \int_D [(\nabla \xi)^2 + (\nabla \eta)^2] dA \quad (2-13)$$

and the **orthogonality** is defined as

$$I_o = \int_D (\nabla \xi \cdot \nabla \eta)^2 J^3 dA \quad (2-14)$$

where the factor J^3 is added to enforce orthogonality with higher weighty value in large cells.

If the three indices approach their minimum values, the mesh would have the highest uniformity, orthogonality and adaptivity. In general, a mesh can be generated by minimizing the sum of the three integrals.

$$I = \lambda_w I_w + \lambda_s I_s + \lambda_o I_o \quad (2-15)$$

Since it is impossible to achieve these three objectives at the same time, for a particular mesh one needs to select the appropriate combination of the coefficients of λ_w , λ_s , and λ_o .

According to Brackbill and Saltzman (1982), the 2-D Euler-Lagrange equations to solve this variational problem are as follows.

$$b_1 x_{\xi\xi} + b_2 x_{\xi\eta} + b_3 x_{\eta\eta} + a_1 y_{\xi\xi} + a_2 y_{\xi\eta} + a_3 y_{\eta\eta} = -J^2 \frac{\lambda_w}{2} w \frac{\partial w}{\partial x} \quad (2-16a)$$

$$a_1 x_{\xi\xi} + a_2 x_{\xi\eta} + a_3 x_{\eta\eta} + c_1 y_{\xi\xi} + c_2 y_{\xi\eta} + c_3 y_{\eta\eta} = -J^2 \frac{\lambda_w}{2} w \frac{\partial w}{\partial y} \quad (2-16b)$$

It is a Poisson equation system and can be solved with any iterative technique. For completeness, the coefficients are reproduced from the cited paper and presented below:

$$\begin{aligned} a_i &= \lambda_s a_{s_i} + \lambda_w a_{w_i} + \lambda_o a_{o_i} \quad (i = 1, 2, 3) \\ b_i &= \lambda_s b_{s_i} + \lambda_w b_{w_i} + \lambda_o b_{o_i} \quad (i = 1, 2, 3) \end{aligned} \quad (2-16c)$$

$$\begin{aligned} c_i &= \lambda_s c_{s_i} + \lambda_w c_{w_i} + \lambda_o c_{o_i} \quad (i = 1, 2, 3) \\ a_{s_1} &= -A\alpha; b_{s_1} = B\alpha; c_{s_1} = C\alpha \\ a_{s_2} &= AB; b_{s_2} = -B\beta; c_{s_2} = C\beta \\ a_{s_3} &= -A\gamma; b_{s_3} = B\gamma; c_{s_3} = C\gamma \end{aligned} \quad (2-16d)$$

where

$$A = x_\xi y_\xi + x_\eta y_\eta, B = y_\xi^2 + y_\eta^2, C = x_\xi^2 + x_\eta^2 \quad (2-16e)$$

and

$$\alpha = \frac{x_\eta^2 + y_\eta^2}{J^3}, \quad \beta = \frac{x_\xi x_\eta + y_\xi y_\eta}{J^3}, \quad \gamma = \frac{x_\xi^2 + y_\xi^2}{J^3} \quad (2-16f)$$

$$a_{w_1} = -x_\eta y_\eta, b_{w_1} = y_\eta^2, c_{w_1} = x_\eta^2$$

$$a_{w_2} = x_\xi y_\eta + x_\eta y_\xi, b_{w_2} = -2y_\xi y_\eta, c_{w_2} = -2x_\xi x_\eta \quad (2-16g)$$

$$a_{w_3} = -x_\xi y_\xi, b_{w_3} = y_\xi^2, c_{w_3} = x_\xi^2$$

$$a_{o_1} = x_\eta y_\eta, b_{o_1} = x_\eta^2, c_{o_1} = y_\eta^2$$

$$a_{o_2} = x_\xi y_\eta + x_\eta y_\xi, b_{o_2} = 2(x_\xi x_\eta + y_\xi y_\eta), c_{o_2} = 2(x_\xi x_\eta + 2y_\xi y_\eta)$$

$$a_{o_3} = x_\xi y_\xi, b_{o_3} = x_\xi^2, c_{o_3} = y_\xi^2 \quad (2-16h)$$

2.4 Laplacian Smoothing Method

Because of the way the algebraic mesh is generated, the global properties of the mesh, such as uniformity, orthogonality, could not be guaranteed to be optimal. Smoothing method could be applied to further improve these generated meshes. The Laplacian smoothing scheme is currently adopted in CCHE2D Mesh Generator. This method can be applied locally to improve local nodal distribution and avoid massive global change.

The Laplacian smoothing method is to move one interior node to the center of the polygon formed by the elements around this node. For any internal node $P_i(x_i, y_i)$, Laplacian polygon smoothing method is expressed by

$$x_i^n = \sum_{k=1}^{N(P_i)} x_k / N(P_i) \quad (2-17a)$$

$$y_i^n = \sum_{k=1}^{N(P_i)} y_k / N(P_i) \quad (2-17b)$$

where $N(P_i)$ is the number of nodes around P_i ; the superscript “ n ” means the new value. Note that the number of nodes around P_i is dependent on the shape of the element. For the quadrilateral elements, the number of nodes around P_i is always 8. (Fig 2-8)

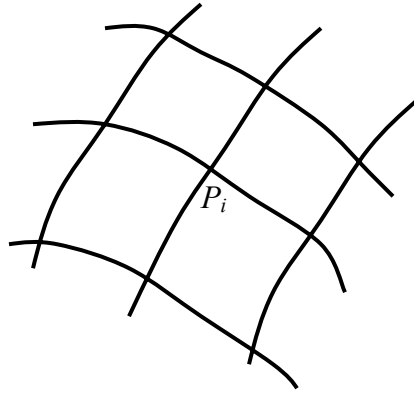


Fig 2-8 Laplacian 8-points-polygon averaged method

Smoothing method is often applied with several iterations. Under-relaxation is used to avoid abrupt change of the nodal location, which could also reduce the mesh distortion near the boundaries caused by smoothing.

$$x_i = (1 - w_i) \cdot x_i + w_i \cdot x_i'' \quad (2-18a)$$

$$y_i = (1 - w_i) \cdot y_i + w_i \cdot y_i'' \quad (2-18b)$$

Where $0 \leq w_i \leq 1$.

2.5 Interpolation of Bed Elevation

Given the basic bathymetry data, there are several methods available for interpolating bed elevation from a database to the mesh points, such as Taylor series expansion method, Lagrange interpolation method, inverse-distance interpolation method, etc.

Random Database

For non-structured bathymetry database, an inverse-distance method is often used in CCHE mesh generator, which requires selecting four surrounding points to complete the interpolation.

The procedure to search these surrounding points is shown in Fig 2-9. First, a mesh point is placed at the origin of a local Cartesian coordinate system. In each quadrant, a point in the bed elevation database closest to this mesh point is selected.

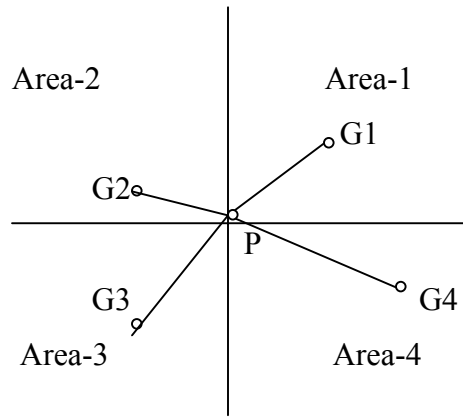


Fig 2-9 Searching Surrounding Points

The interpolation is performed to obtain the bed elevation at the mesh point according to

$$m(x, y) = \frac{\sum \frac{g_i(x, y)}{d_i^n}}{\sum \frac{1}{d_i^n}} \quad (2-19)$$

where $m(x, y)$ is the bed elevation of the mesh point;

$g_i(x, y)$ is the bed elevation of the surrounding points;

d_i is the distance between each surrounding point and the interpolated mesh point;
 n is the power.

Structured Database

If the structured survey database is available, in order to improve the accuracy of the interpolation, a refined-database is created considering that channel thalweg should be connected.

Creating such a refined database involves the following steps.

- (1) Refine the data points in the transverse direction along each cross section using linear interpolation and make the number of points on each cross section equal to $2 \times N_{pmax}$. N_{pmax} is the maximum number of points of the cross section in the original database. This is the first refinement.
 - (2) Find the thalweg point of each refined cross section.
 - (3) The thalweg point divides each cross section into two parts. In each part, use the linear interpolation to create N_{pmax} data points. This is the second refinement.
 - (4) According to the latest database, linear interpolation is used to create data points between two cross sections. This is the last refinement.
 - (5) Based on the refined database, use the inverse-distance method to interpolate the bed elevation.
-

Chapter 3

Interface of CCHE2D Mesh Generator

In order to generate computational mesh efficiently with the CCHE2D Mesh generator, a Graphic Users' Interface (GUI) is developed. This powerful interface is designed to be user-friendly, interactive and efficient.

3.1 Introduction of Interface

Fig 3-1 shows the GUI of CCHE2D Mesh Generator. It is composed of a big graphic window and operations for mesh generation which are grouped into several categories.

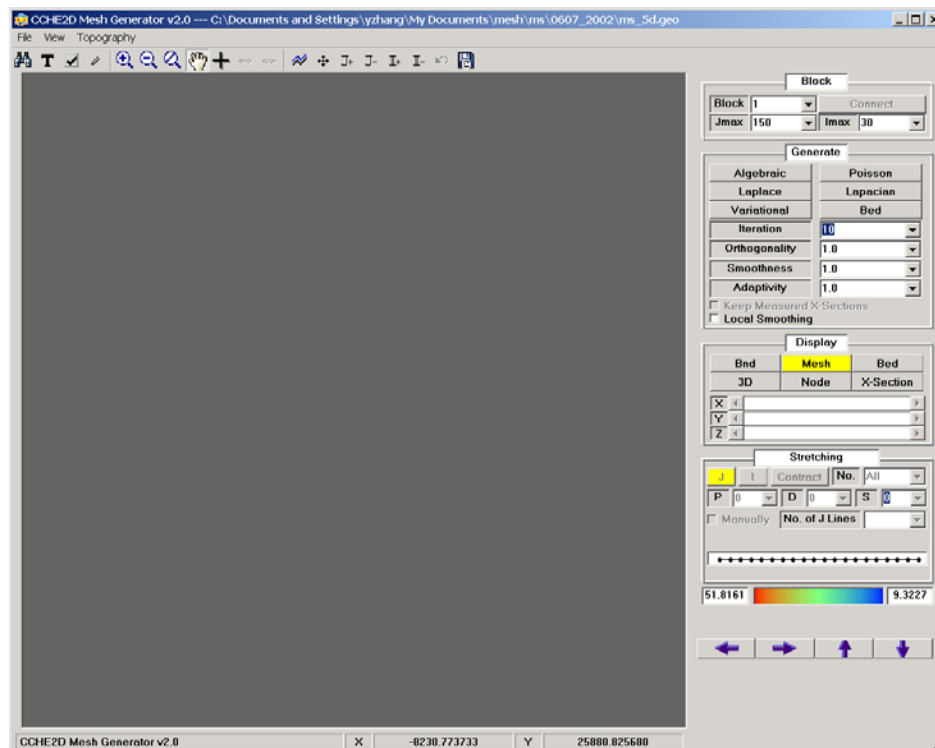


Fig 3-1 Interface of CCHE2D Mesh Generator

3.1.1 Groups of Main Window

As Fig 3-1 shows, almost all the commands, the inputting parameters and the labels are placed group by group on the front face of the main window for easy access. These groups of commands and parameters are introduced as follows.

1. **“Block” group:** In this group, the user can select the block (sub-domain) and specify the mesh size ($I_{\max} \times J_{\max}$) for this block (sub-domain). After finishing mesh generation for several blocks (sub-domains), click “Connect” button and all the generated meshes are linked into one big mesh.

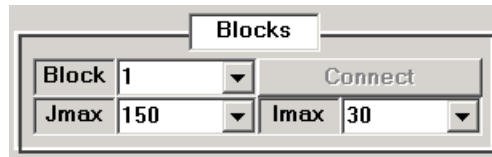


Fig 3-2 “Block” Group

2. **“Generate” group:**

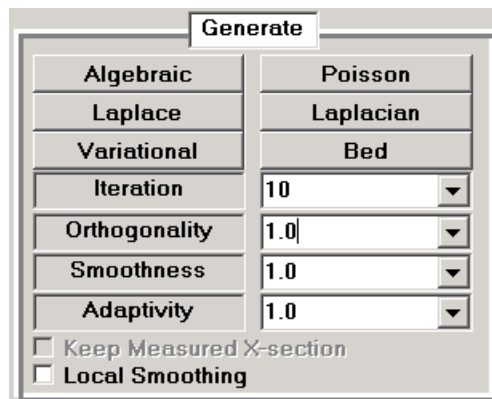


Fig 3-3 “Generate” Group

This group is a collection of commands to generate a mesh, including six command buttons, four inputting boxes and two check boxes. Once a command button is clicked, an

action of the mesh generation is activated and this button will turn into yellow to provide a friendly reminder.

- Click “Algebraic” button and the algebraic mesh will be obtained. Usually, this is the first button one needs to push when generating a mesh.
- Click “Poisson” button and the Poisson Equation for orthogonal mapping will be solved.
- Click “Variational” button, the Variational Laplace Equation will be solved to obtain adaptive mesh.
- Click “Laplacian” button and the mesh will be smoothed using Laplacian polygon averaged method.
- Click “Laplace” button and the Laplace Equation for conformal mapping will be solved.
- Click “Bed” button to interpolate the bed elevation based on the survey database. Details will be covered in later section.

The “Orthogonality”, “Smoothness”, and “Adaptivity” parameters are required for both Poisson and Variational method and the “Max Iteration” parameter is required for all the methods except the algebraic method. All these parameters have default values. Note that for the smoothing methods the “Max Iteration” parameter is recommended to be less than 10 to avoid the distortion caused by excessive smoothing, especially for complicated cases, because “Smoothing” can be performed repeatedly and the user can improve the mesh gradually.

3. **“Display” group**: This group provides different ways to view mesh. In this group, there are six commands: “Bnd”, “Mesh”, “Bed/ Grided Bed”, “3D”, “Node”, and “X-section”; and three scroll bars for 3D view: “X”, “Y”, and “Z”.

- Click “Bnd”, the boundary will be displayed.
- Click “Mesh”, the mesh of the selected block will be shown.

- Click “Bed” once, the color-shaded bed form will be displayed and the caption of this button will turn into “Grided Bed”; Click this button again, the color-shaded bed with mesh on will be displayed and the caption of this button will turn into “Bed”.
- Click “3D”, the 3-dimensional geometry of the domain will be shown and the three scroll bars will be activated. The user can rotate the 3D topography with these scroll bars.
- Click “Node”, the mesh nodes colored according to bed elevation will be displayed.
- Click “X-section”, a sub-window will pop out and the cross-sections will be displayed. Details will be covered in later section.

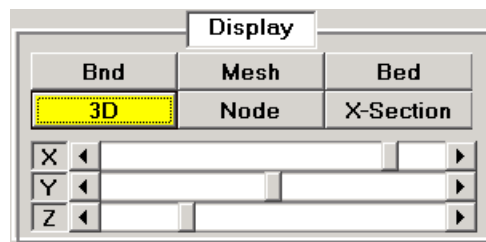


Fig 3-4 “Display” Group

4. “**Tool Bar**” group: This is a collection of tools for editing.



Fig 3-5 “Edit” Group

Each button is represented by a graphical icon and they can be grouped into three functional categories: “Mesh Information”, “Mesh Editing”, and “Mesh Viewing”.

- **Mesh Information:**

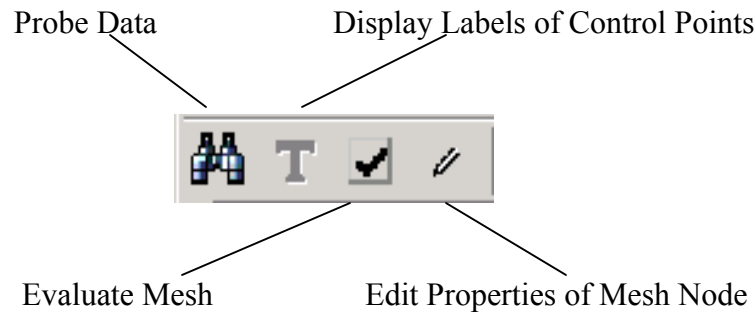


Fig 3-6 Tools for Mesh Information

The “Probe Data” button provides nodal information. When the mouse moves on the mesh face, the user can get the local mesh information simultaneously. The local information displayed is: (I, J) indices, (G) global number of node, (X, Y, Z) coordinates, (I.D) nodal type, and (n) Manning’s Roughness n .

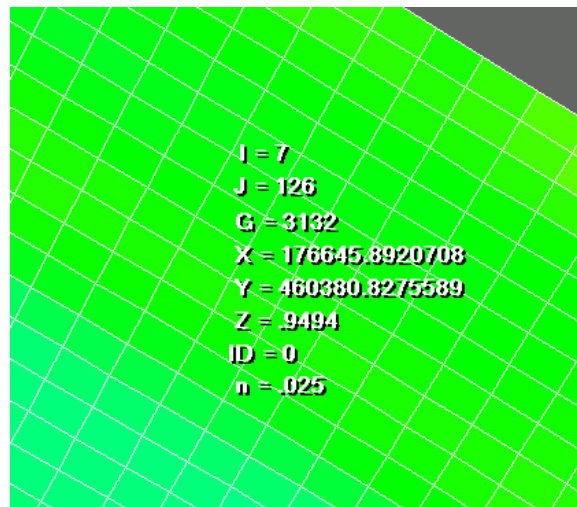


Fig 3-7 Local information window

Click “Evaluate Mesh” button and the user can get a brief evaluation of the mesh. Currently, two indicators of mesh quality are identified: “Smoothness” and “Orthogonality”.

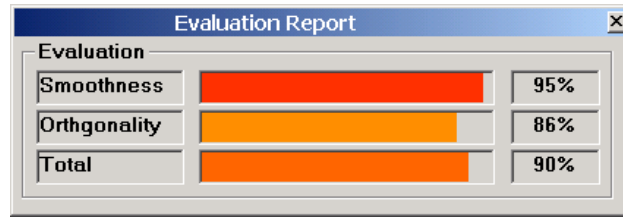


Fig 3-8 Evaluation report window

Click “Display Labels of Control Points” and the control sections will be marked.

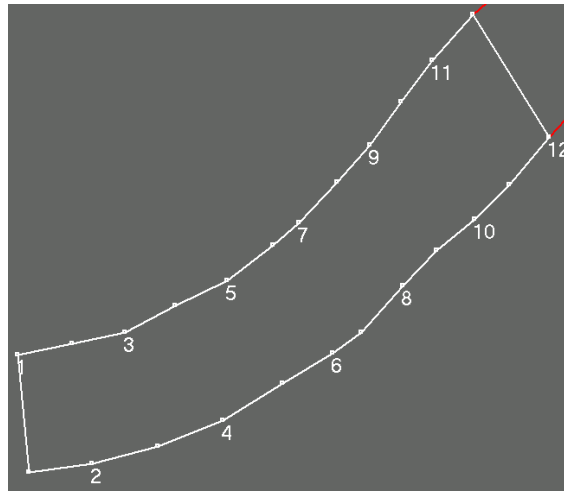


Fig 3-9 Display Labels of Control Points

Click “Edit Properties of Mesh Nodes” button, an edit-pen window will pop out and the user can modify the local information of the mesh. Details will be covered in later section.

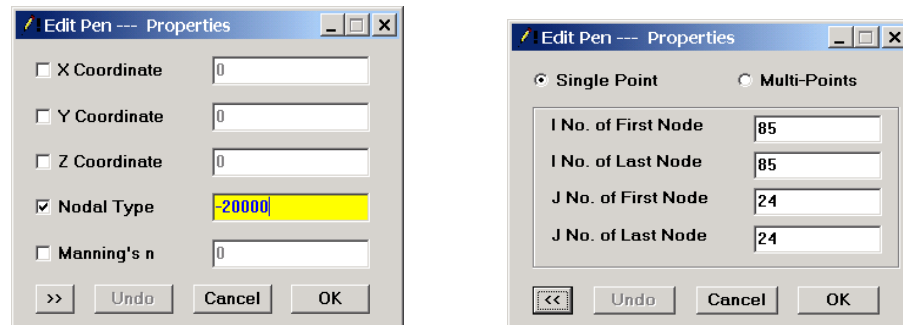


Fig 3-10 Edit-pen window

- **Mesh Editing:**

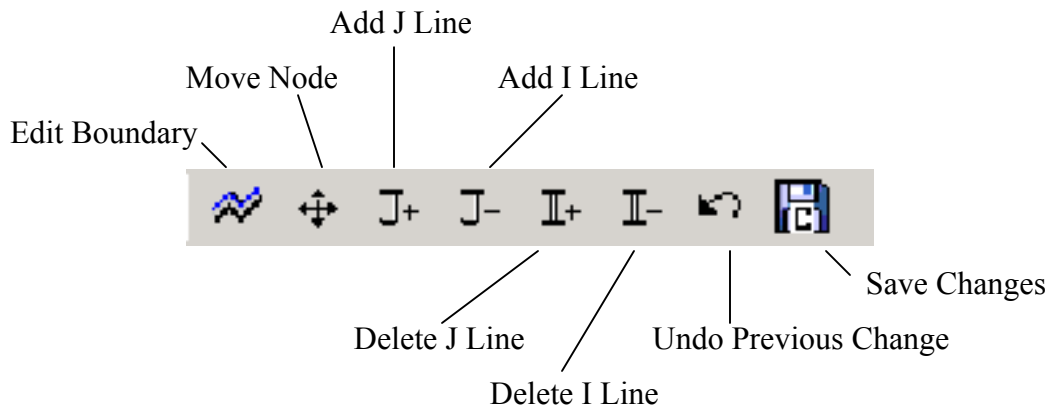


Fig 3-11 Tools for Mesh Editing

- **Mesh Viewing:**

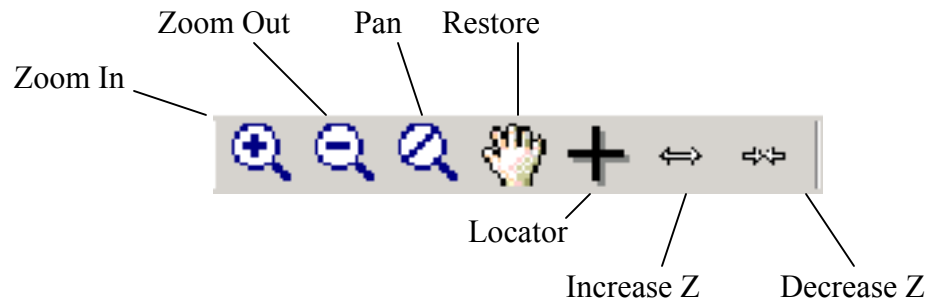
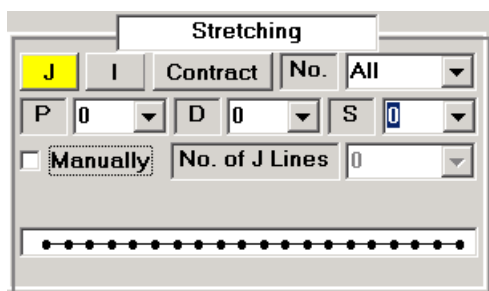
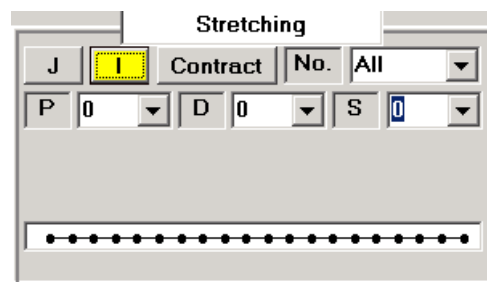


Fig 3-12 Tools for Mesh Viewing

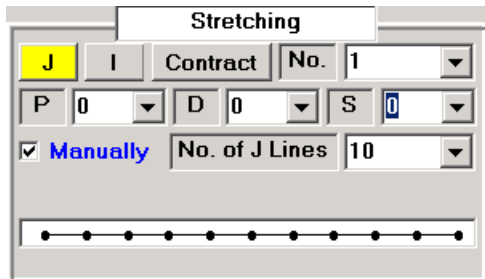
5. **“Stretch Function” group:** In this group, the stretching function parameters are set.



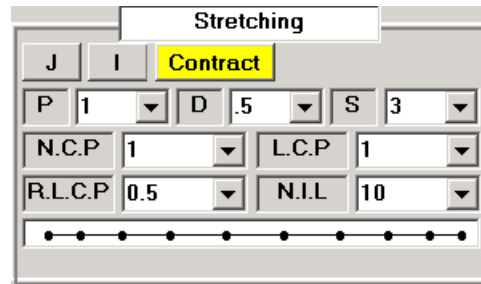
(a) Click button “J”



(b) Click button “I”



(c) Check “Manually”



(d) Click button “Contract”



Pattern of the nodal distribution according to current “P”, “D”, “S”.

Fig 3-13 “Stretch Function” Group

This is the most complicated group. It provides three main functions: set the stretching parameters in each sub sections; set the stretching parameters for I lines in the transverse direction; and set the contraction or repulsion points along I lines.

When the user click the options and the check box, the related inputting boxes will turn to same color as shown in Fig 3-13.

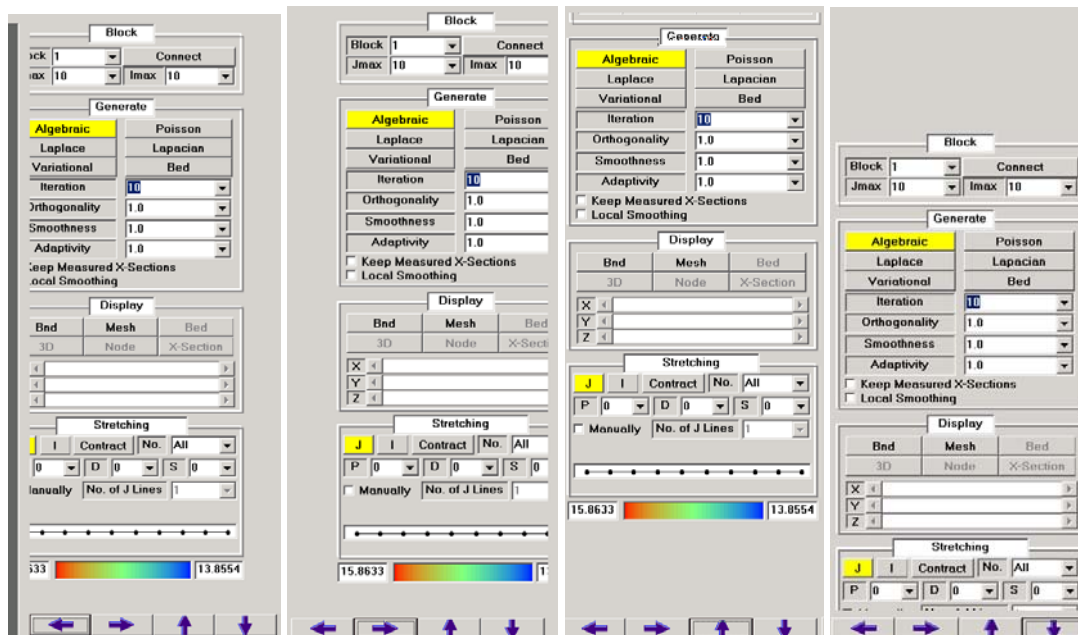
- “Bnd” means “Boundary lines of each subsection”;
- “I line” means “Mesh I lines”;
- “Manual distribution” means “Distribute mesh lines for each subsection manually”;
- “N.I.L” means “No. of mesh **I** Lines distributed in the selected subsection”;
- “N.C.P” means “No. of **C**ontraction or repulsion **P**oints”;
- “L.C.P” means “**L**abel of each **C**ontraction or repulsion **P**oint”;
- “R.L.C.P” means “**R**elative **L**ocation of **C**ontraction or repulsion **P**oint”;
- “N.J.L” means “No. of **J** Lines distributed in each contracted or repulsed section”.

3.1.2 Control Panel

In current version of CCHE2D Mesh Generator, a control panel is newly added into the interface, which is designed to adjust the location of operation controls on the interface. Under the low resolution, for example, 800×600 , some controls may not visible and then this control panel can be used to pull out those invisible controls and make them visible.



- Click , all the controls move leftward with the same distance.
- Click , all the controls move rightward with the same distance.
- Click , all the controls move upward with the same distance.
- Click , all the controls move downward with the same distance.



(a) Move leftward (b) Move rightward (c) Move upward (d) Move downward

Fig 3.14 Control Panel

3.1.3 Menus

There are three menus in the main window: “File” menu, “View” menu, and “Bed Elevation” menu. “File” menu provides the operation of the files while “View” menu provides the view tools of the picture. “Bed Elevation” menu provides functions related to bed elevation.

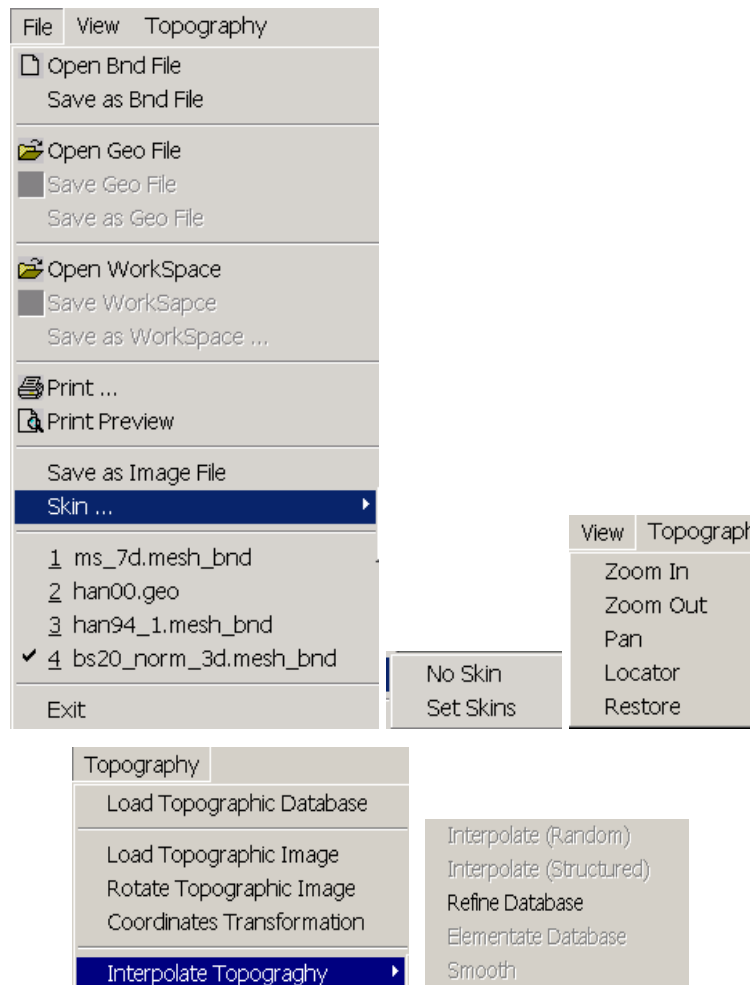
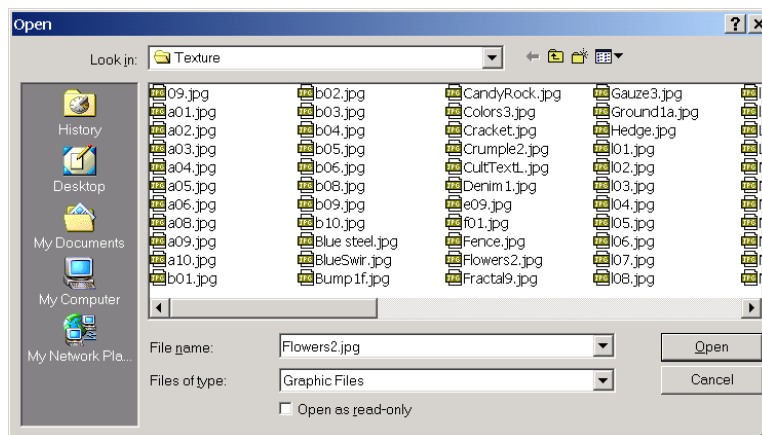


Fig 3-15 Menus

“File” Menu:

There are files history displayed in the “File” menu and the user can click them and open them directly.

- Click “Open Bnd File”, a boundary file that defines the control points along the domain boundaries will be opened and the domain will be drawn. (*.mesh_bnd)
- Click “Save as Bnd File”, a boundary file that defines the control points along the domain boundaries will be saved. (*.mesh_bnd)
- Click “Open Geo File”, a geo file of an existing mesh will be opened. (*.geo)
- Click “Open WorkSpace”, a mesh work-space file will be opened. (*.mesh_wsp)
- Click “Save WorkSpace” and the current workspace will be saved.
- Click “Save as WorkSpace”, the mesh with all its parameters will be saved as a mesh work-space file;
- Click “Save Geo File”, the current mesh will be saved as a geo file.
- Click “Save as Geo File”, the current mesh will be save as another geo file.
- Click “Print” and the current picture will be printed out.
- Click “Print Preview” and a preview window will pop out.
- Click “Save as Image File” and the picture drawn on the screen will be saved as a bitmap file.
- Click “Skin” and a submenu will be shown where there are two menus, “No skin” and “Set Skin”. “Set skin” allows the user to choose a picture as the background for the main window; and “No skin” will clean the background.



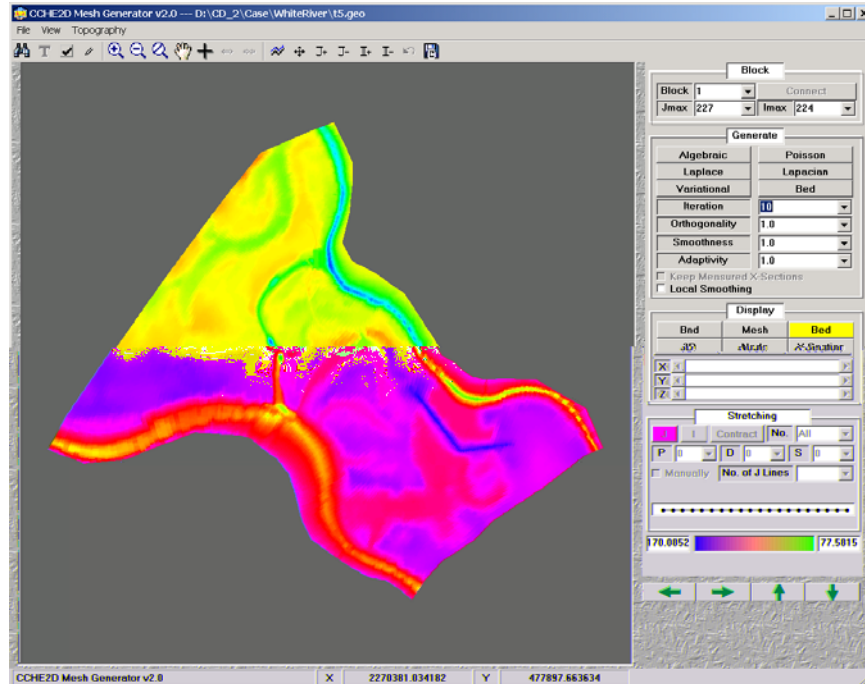


Fig 3-16 Set Skin

- Click “Exit” and the user will quit the mesh generator.

“View” Menu:

- Click “Zoom In” and the user can specify one area he wants to see in detail.
- Click “Zoom Out” and the user can zoom out the picture.
- Click “Pan” and the user can move the picture.
- Click “Locator” and two cross lines will be shown whose intercross point moves with the mouse.
- Click “Restore” and the picture will be restored into full-screen state.

“Topography” Menu

This menu consists of two levels. The first level is main menu and the components are as follows:

- Click “ Load Geometry Database”, a geometry database file will be opened and the survey points colored due to bed elevation will be shown. (*.mesh_xyz)

- Click “Load Topographic Image” and an image file (*.bmp) will be opened.
- Click “Coordinates Transformation” and a window will pop out which is used to transform the local coordinate of the screen into the real coordinate of the topographic map.

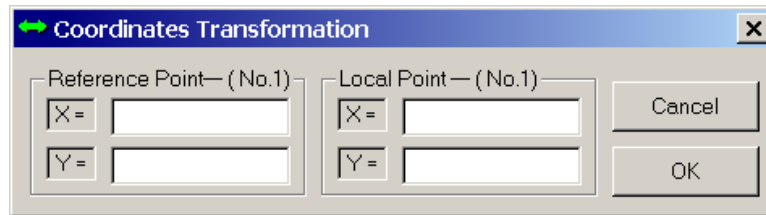


Fig 3-17 Coordinates Transformation

The second level menu is the sub-menus of “Interpolate Topography”.

- Click “Interpolate (Random) ” and the bed interpolation based on the random survey database will begin.
- Click “Interpolate (Structured) ” and the bed interpolation based on the structured survey database will begin.
- Click “ Refine Database” and the structured survey database will be refined.
- Click “Elementate Database” and the structured survey database will be discretized into elements (triangles and quadrilaterals).
- Click “Smooth” and the user can smooth the bed elevation of the specified area.

3.2 How to Generate a Mesh

3.2.1 Generate Mesh from Topographic Database File

At the beginning, the topographic database is the only file available. This database file can be used not only as the interpolation database but also to create a boundary file required for mesh generation. An example will be shown how to make a boundary file using geometry database file. Note the geometry database file has the extension “mesh_xyz”.

Boundary Creator

The boundary creator plays an important role during this process and it will be introduced in detail as follows.

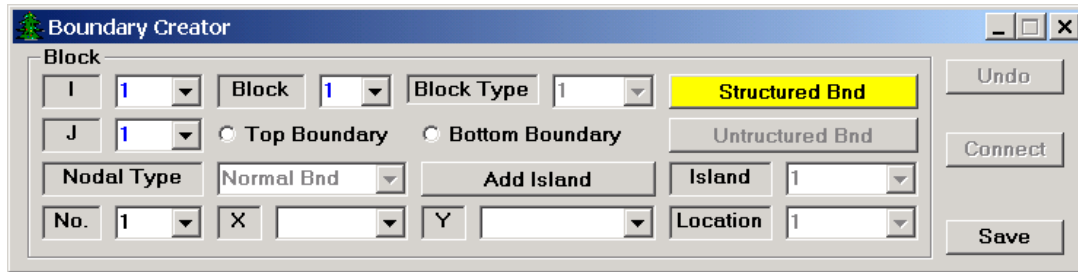


Fig 3-18 Boundary Creator Window

In this window, there are three groups: “Structured Bnd” group, “Unstructured Bnd” group and “Island” group. In current version, the “Unstructured Bnd” group is disabled.

Structured Bnd

The multi-block scheme is applied in CCHE2D mesh generator and the “Structured Bnd” means that the blocks are arranged in a same way as the structured mesh: Apart from a global number, each block has the (I, J) indices to be uniquely identified.

The user needs to click “Structured Bnd” button to activate this group. This group provides the operations to create a structured multi-block boundary. In this group, there are six inputting boxes and two options and they are:

- “I” and “J” are the indices of the current block
- “Block” is the global number of the current block
- Option “Top Boundary” means that the operation is worked on the top boundary of the current block.
- Option “Bottom Boundary” means that the operation is worked on the bottom boundary of the current block.

- “No.” records the number of control points of the current block.
- “X” is the x-coordinate of the control point of the current block.
- “Y” is the y-coordinate of the control point of the current block.

Island

In each block, the user can add islands into it, which is provided by “island” group. The user needs to click “Add Island” to activate this group and switches from block operation to island operation. Note that the island can be added into only the completed block. If the block is not completed, an error message will be given.

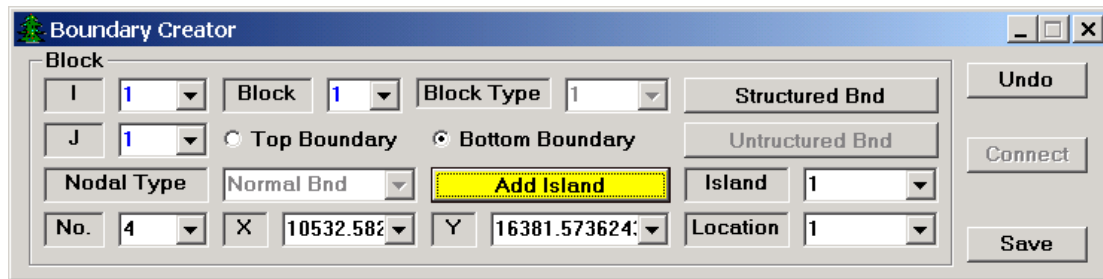


Fig 3.19 Click “Add Island”

There are five inputting boxes and two options in this group and they are:

- “Island” records the number of islands in current block.
- “Location” is the relative location of the first pair of control points of the current island in current block.
- Option “Top Boundary” means that the operation is worked on the top boundary of the current island.
- Option “Bottom Boundary” means that the operation is worked on the bottom boundary of the current island.
- “No.” records the number of control points of the current island.
- “X” is the x-coordinate of the control point of the current island.

- “Y” is the y-coordinate of the control point of the current island.

There are three command buttons in this boundary creator window. Currently, “Connect” is not enabled.

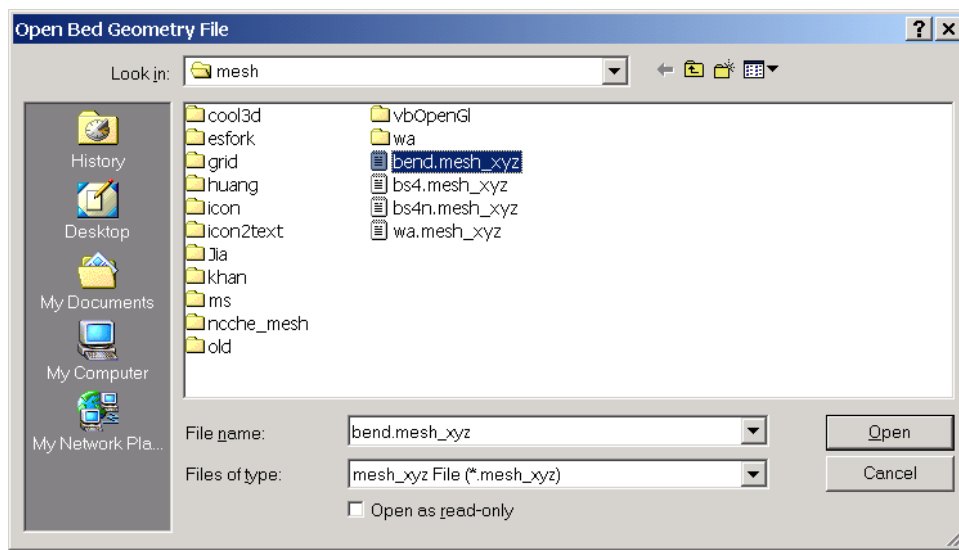
- Click “Undo” and the user can cancel the current action.
- Click “Save” and the boundary file will be created and saved into a file.

In current version, two new features are added into the boundary creator:

- Reminder of top and bottom boundary: in the first block, two labels are provided to remind the user which boundary he is working on.
- Reminder of the directions of I and J of the blocks.

Now an example will be discussed in detail to demonstrate this process.

Step-1 Click “Topography” menu and choose “Load Topographic Database File”. A bend channel is loaded and it will be divided into three blocks in which there is one island.



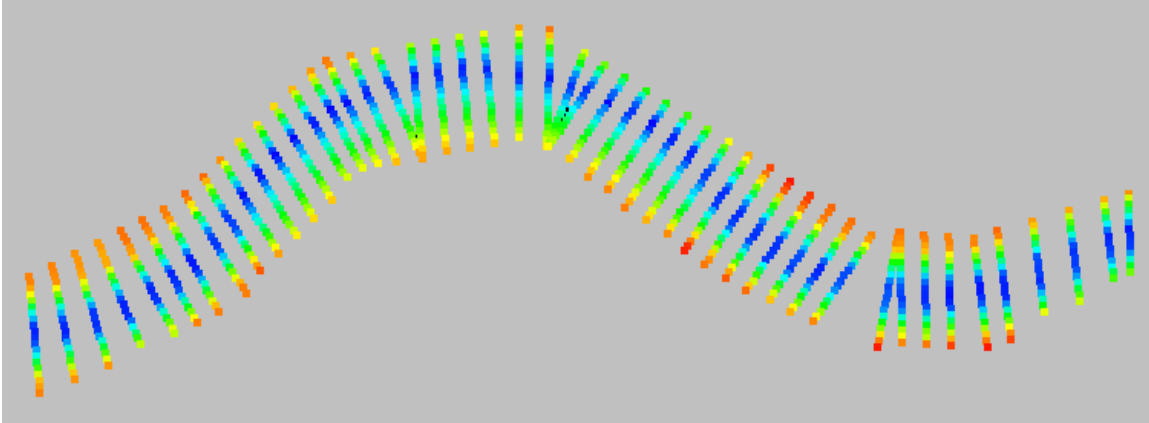
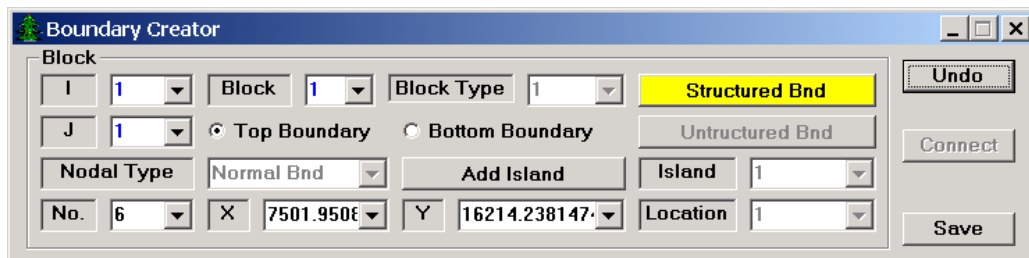


Fig 3-20 Load Geometry Database File

Step-2 Click “” button in “Edit Tool” group and the boundary creator will pop out.

Step-3 Specify the control points for the first block.

- (1) Evaluate block (I, J) values as (1,1).
- (2) Choose option “Top Boundary”.
- (3) Press “Ctrl” key and hold it.
- (4) Move the mouse pointer to the desired location and press right button. Thus one control point is specified.
- (5) Repeat this process until all the control points on the top boundary are specified.



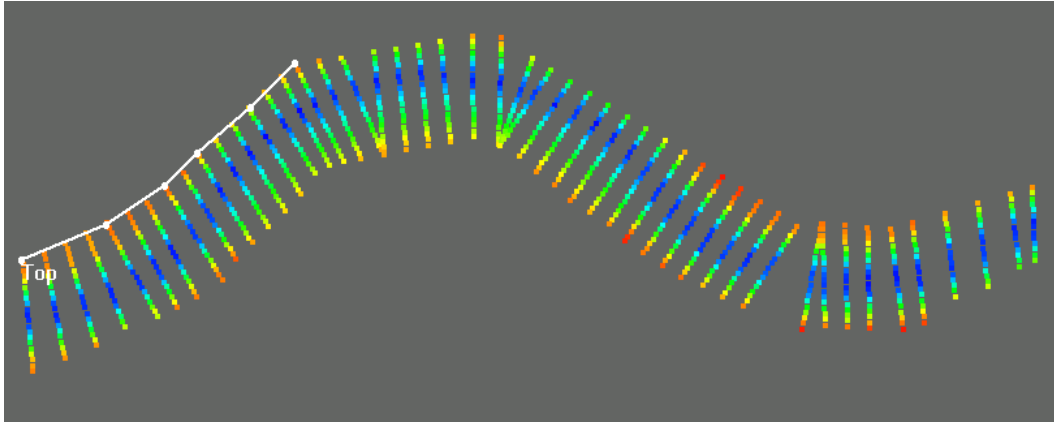
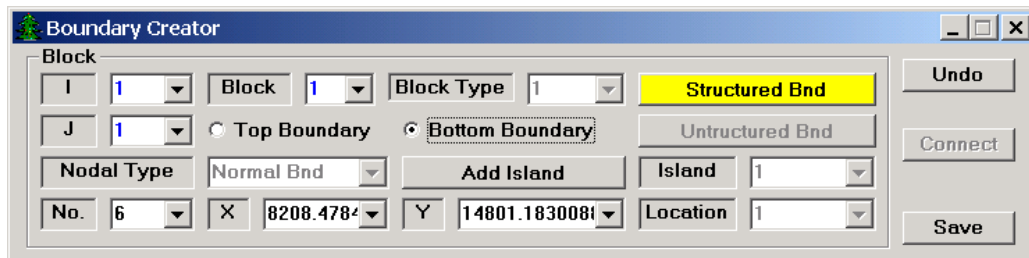


Fig 3-21 Specify control points on top boundary of the first block

- (6) Choose option “Bottom Boundary” and apply the similar operations. Note that the number of control points on the bottom boundary must be equal to the number of points on top boundary.



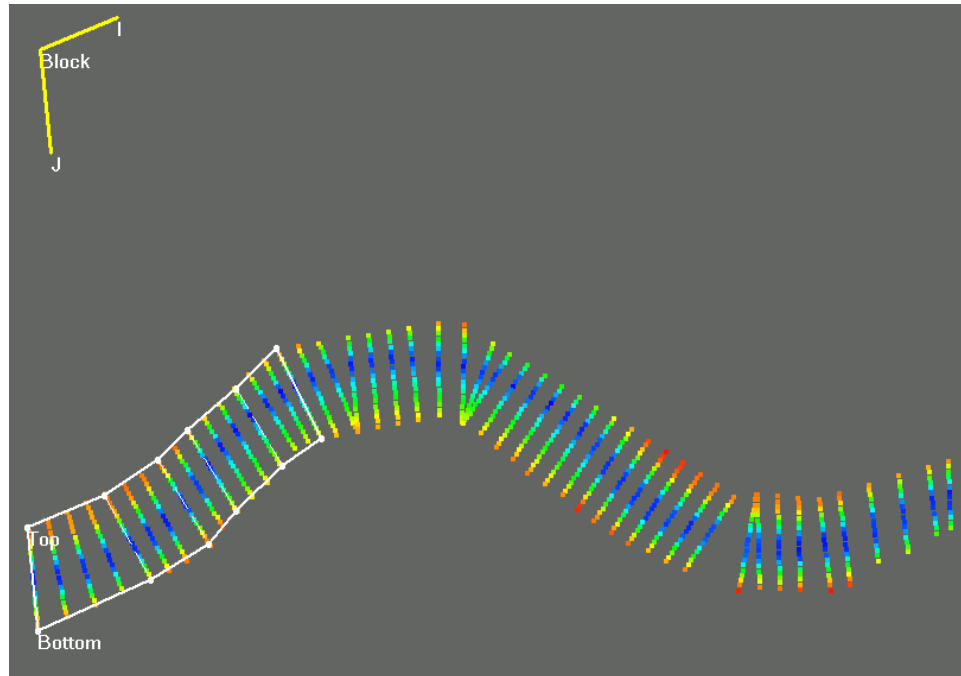
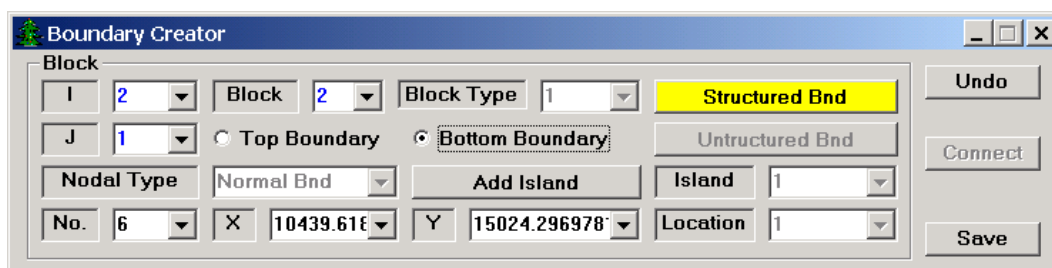


Fig 3-22 Specify control points on bottom boundary of the first block

- (7) Make sure all the parameters of the first block specified and then move onto the next block

Step-4 Specify the control points for the second block and note an island is required to be added into this block.

- (1) Evaluate the block (I, J) values as (2, 1) and then do the same operations as block 1.



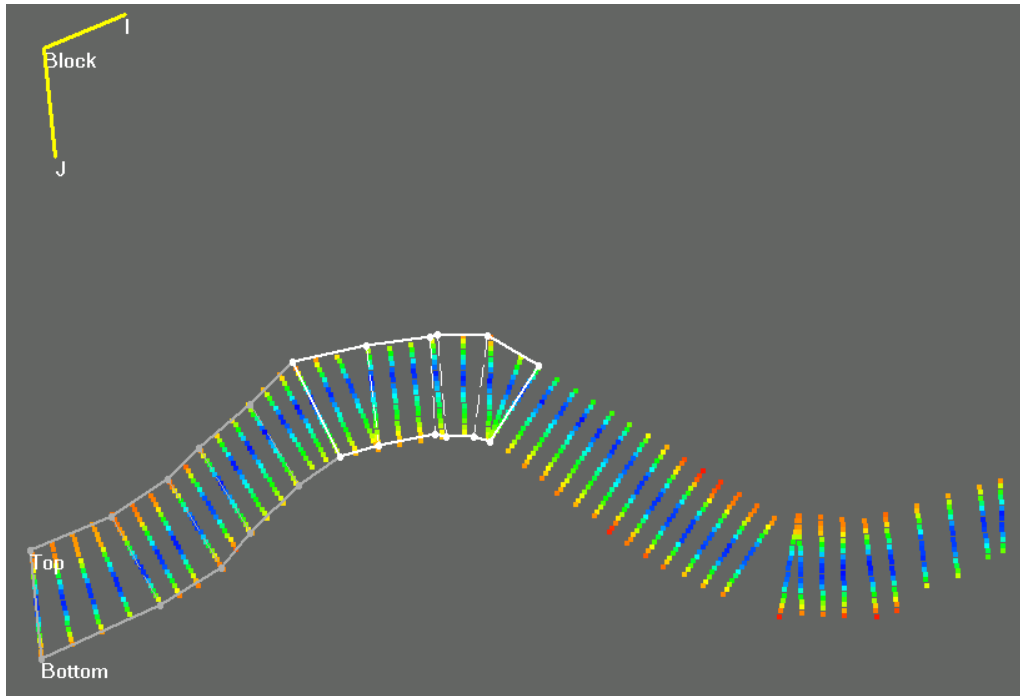
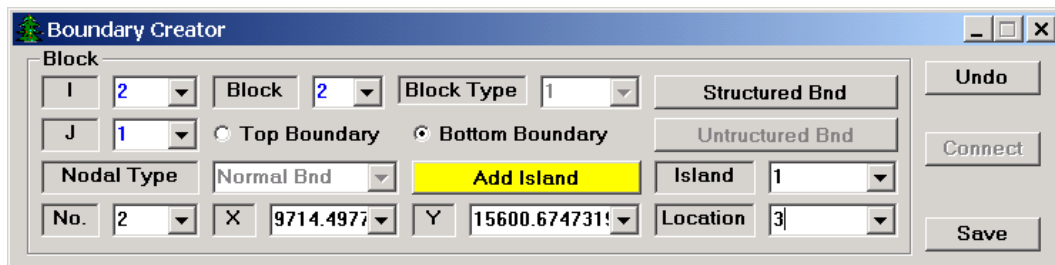


Fig 3-23 Specify the control points for block 2

- (2) After this block is completed, click “Add Island”. Choose “Top Boundary” and then do the same operations as block to specify the control points of top boundary of the island.
- (3) Choose “Bottom Boundary” to specify the control points of bottom boundary of the island.
- (4) Evaluate the “Location” as “3”.



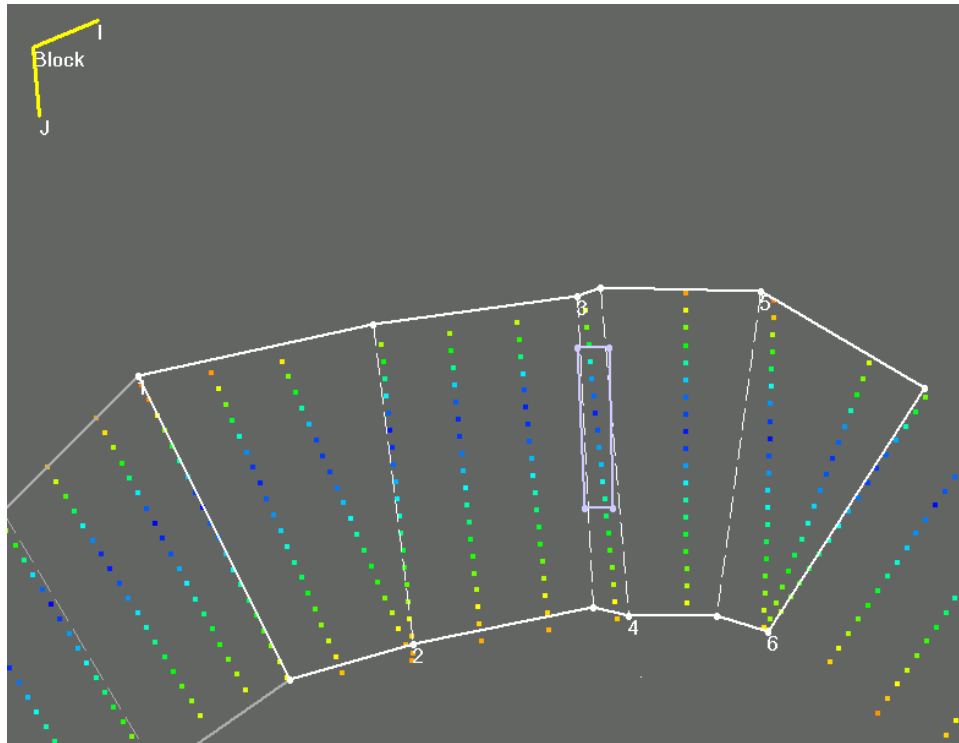
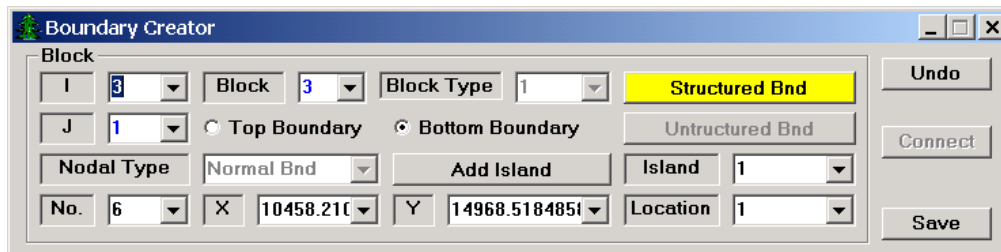


Fig 3-24 Add island into block 2

Step-5 Specify the control points for the third block. First click “Structured Bnd” to switch from island operation to block operation. Evaluate the block (I, J) values as (3, 1) and then do the same operations as block 1.



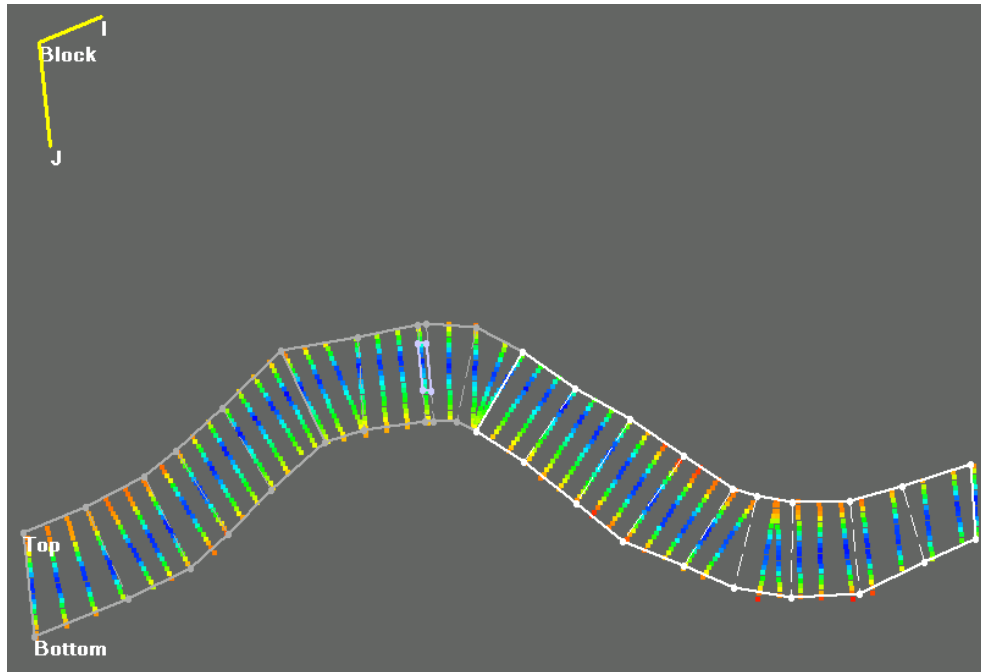


Fig 3-25 Specify the control points for block 3

Step-6 Click “Save” and save all the information as a boundary file.

3.2.2 Generate Mesh from Topographic Image

If the topographic image is available, the user can generate mesh directly from it. The procedure is almost the same as the previous section.

Step-1: Click “Topography” menu and choose “Load Topographic Image”.

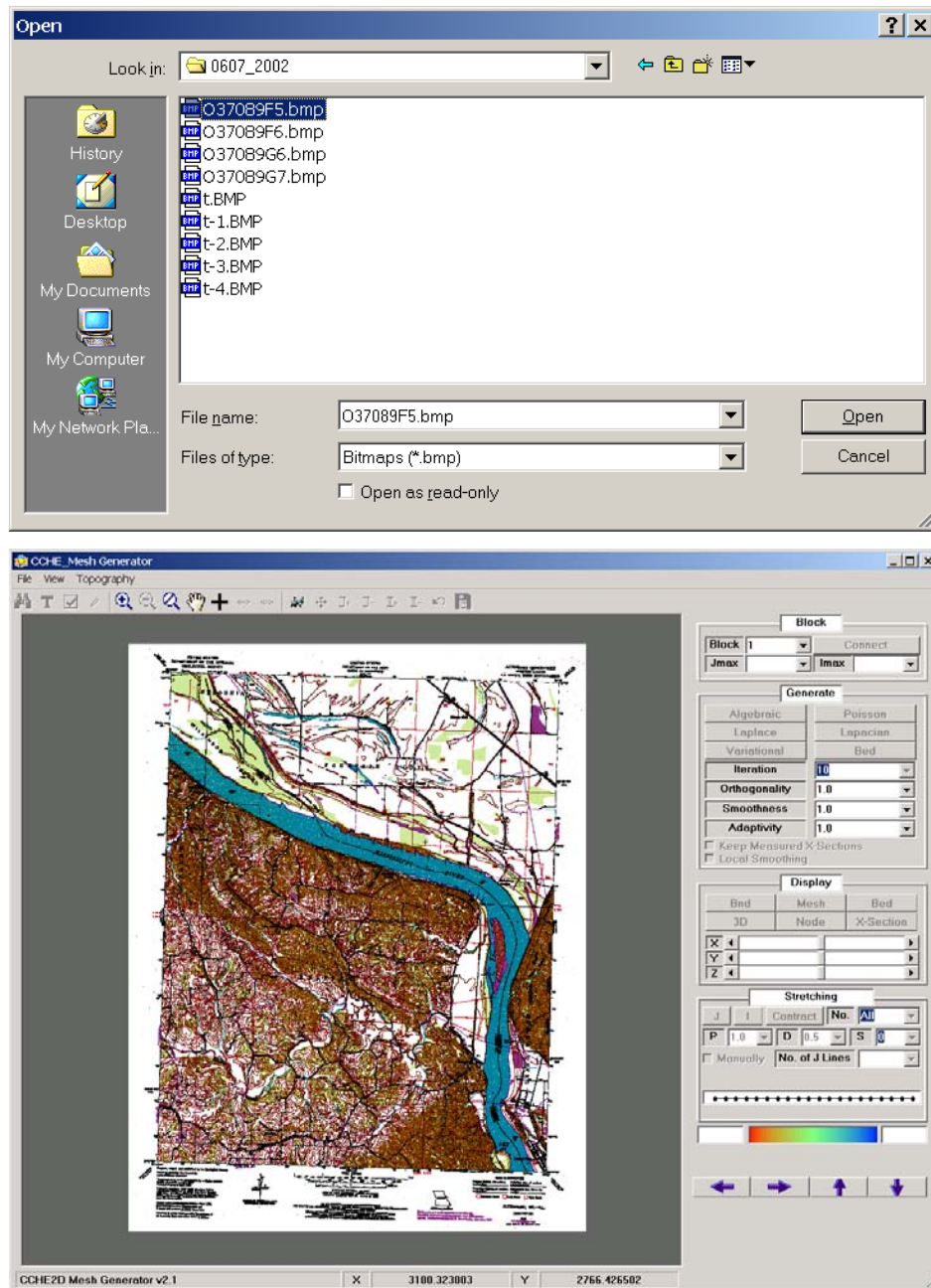


Fig 3-26 Load the Topographic Image

Step-2: Click “Boundary” button in “Edit” group and create the boundary with the help of “Edit Boundary” window.

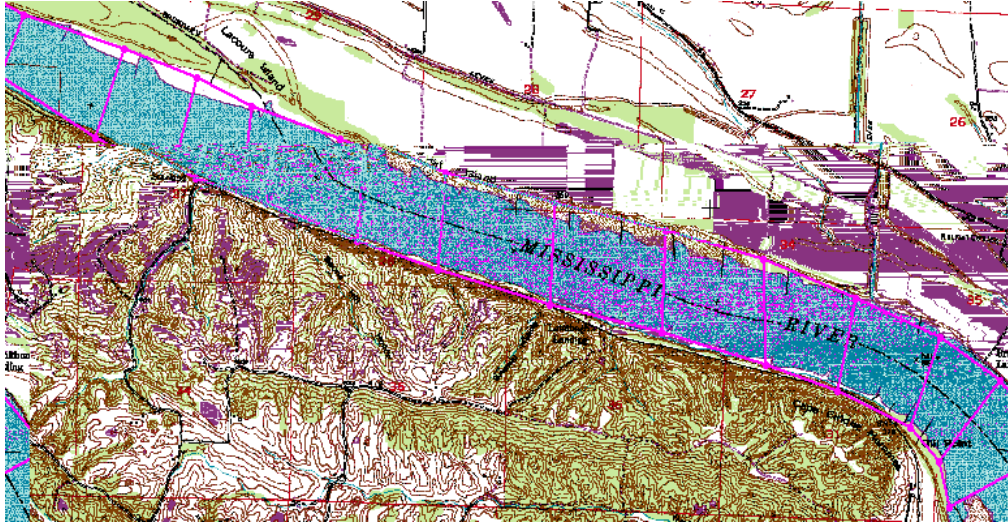


Fig 3-27 Create Boundary

Step-3: Transform the local coordinates system of the screen into the real coordinate system of the topographic map. Click “Topography” menu and choose “Coordinates Transformation”. The user can make the transformation as follows: (1) click the first reference point on the topographic map and input the real coordinates of this points; (2) click the second reference point on the topographic map and input the real coordinates of this points; (3) click “OK” button of the “Coordinates Transformation” window and the transformation parameter will be shown.

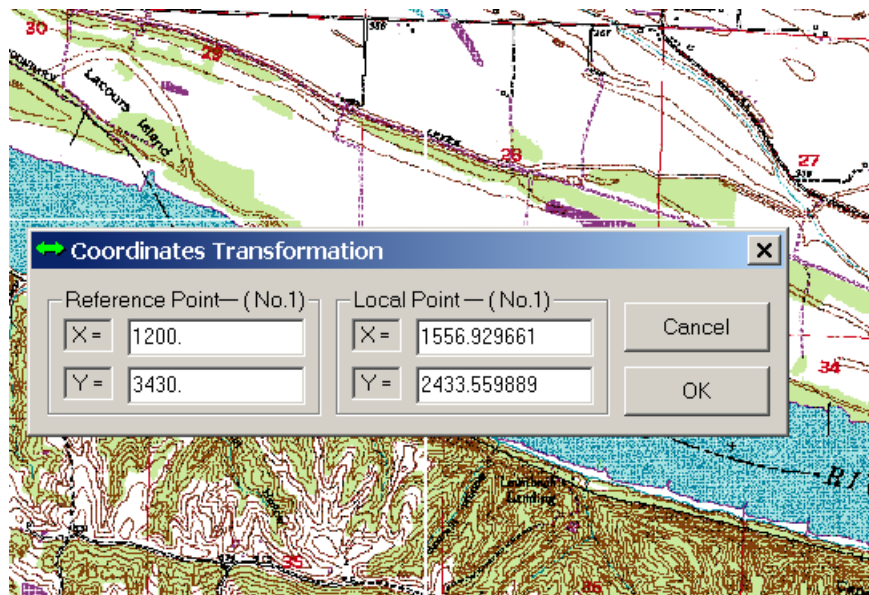


Fig 3-28 Click the First Reference Point

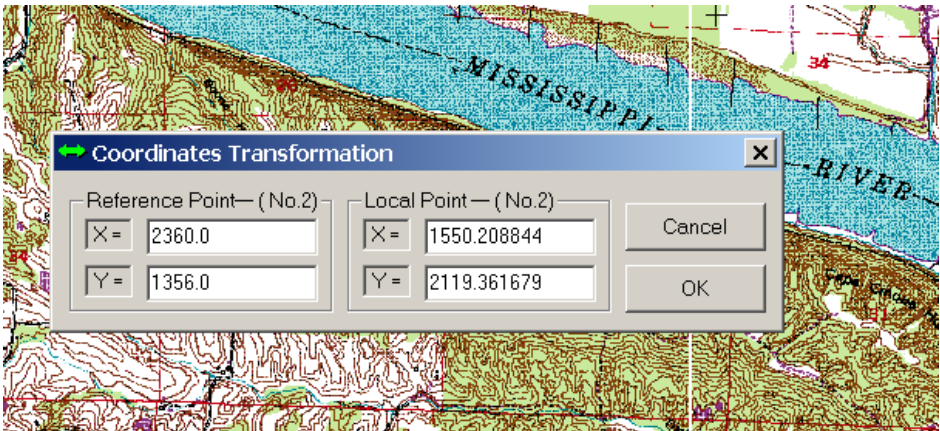


Fig 3-29 Click the Second Reference Point

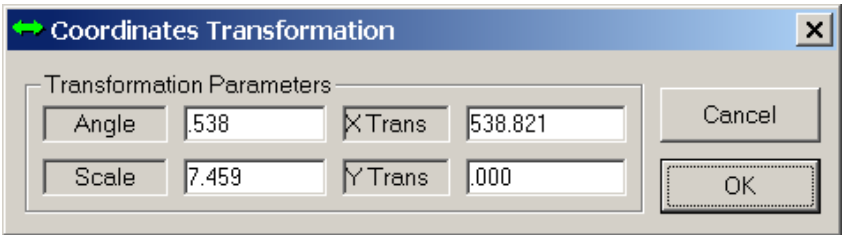


Fig 3-30 Transformation Parameters

Step-4 Click “Save” and save all the information as a boundary file.

3.2.3 Generate Mesh from Boundary File

Basically, if the boundary file is ready and the default values of the parameters are used, it’s very easy to generate a mesh. What a user needs to do is to evaluate the mesh size ($I_{\max} \times J_{\max}$) and then click “Algebraic” button, then the user can get an algebraic mesh.

CCHE mesh generator can only identify the boundary file with an extension “mesh_bnd”. The format of this boundary file is shown in Tab 3-1.

Tab 3-1 Format of Boundary File

3	1	9	Max I No. of Blocks	Max J No. of Blocks	Max No. of Control Points of Blocks
---	---	---	---------------------	---------------------	-------------------------------------

0	1	0	Max No. of straight Dikes		Max No. of Islands		Max No. of Compound Dikes × 2		
6	No. of Pairs of Control Points of (1,1) Block								
0	1	0	1	Four Bnds i.d : left, downward, right, upward (“1”-wall bnd; “0”-not a bnd)					
1	25.71	58.18	27.68	55.01	No.	Xdown	Ydown	Xup	Yup (1 st pair of control points)
2	26.41	58.79	28.31	55.71	No.	Xdown	Ydown	Xup	Yup (2 nd pair of control points)
3	27.40	59.56	29.27	56.54	No.	Xdown	Ydown	Xup	Yup (3 rd pair of control points)
4	28.30	60.24	30.07	57.06	No.	Xdown	Ydown	Xup	Yup (4 th pair of control points)
5	29.13	60.63	30.79	57.40	No.	Xdown	Ydown	Xup	Yup (5 th pair of control points)
6	30.56	61.22	31.90	57.94	No.	Xdown	Ydown	Xup	Yup (6 th pair of control points)
0	No. of straight dikes								
0	No. of Islands								
0	No. of Compound dikes × 2								
6	No. of Pairs of Control Points of (2,1) Block								
0	1	0	1	Four Bnds i.d : left, downward, right, upward (“1”-wall bnd; “0”-not a bnd)					
1	30.56	61.22	31.90	57.94	No.	Xdown	Ydown	Xup	Yup (1 st pair of control points)
2	32.96	62.05	33.33	58.38	No.	Xdown	Ydown	Xup	Yup (2 nd pair of control points)
3	36.13	62.42	34.86	58.60	No.	Xdown	Ydown	Xup	Yup (3 rd pair of control points)
4	36.18	62.42	34.91	58.61	No.	Xdown	Ydown	Xup	Yup (4 th pair of control points)
5	38.02	62.39	38.03	58.71	No.	Xdown	Ydown	Xup	Yup (5 th pair of control points)
6	39.82	62.09	39.69	58.46	No.	Xdown	Ydown	Xup	Yup (6 th pair of control points)
0	No. of straight dikes								
1	No. of Islands								
2	No. of pairs of control points of island								
1	3	36.14	62.42	35.51	60.53	No.	Location	Xdn	Ydn Xup Yup (1 st)
2	4	36.17	62.42	35.55	60.54	No.	Location	Xdn	Ydn Xup Yup (2 nd)
0	No. of Compound dikes × 2								
9	No. of Pairs of Control Points of (3,1) Block								
0	1	0	1	Four Bnds i.d : left, downward, right, upward (“1”-wall bnd; “0”-not a bnd)					
1	39.82	62.09	39.69	58.46	No.	Xdown	Ydown	Xup	Yup (1 st pair of control points)
2	41.65	61.62	40.59	58.27	No.	Xdown	Ydown	Xup	Yup (2 nd pair of control points)
3	43.82	60.49	41.87	57.50	No.	Xdown	Ydown	Xup	Yup (3 rd pair of control points)
4	45.65	59.38	43.69	56.44	No.	Xdown	Ydown	Xup	Yup (4 th pair of control points)
5	46.63	58.64	44.63	55.74	No.	Xdown	Ydown	Xup	Yup (5 th pair of control points)
6	48.10	57.51	45.88	54.38	No.	Xdown	Ydown	Xup	Yup (6 th pair of control points)
7	49.33	56.10	47.36	53.38	No.	Xdown	Ydown	Xup	Yup (7 th pair of control points)
8	50.08	55.05	48.26	52.51	No.	Xdown	Ydown	Xup	Yup (8 th pair of control points)
9	51.45	53.83	49.69	51.32	No.	Xdown	Ydown	Xup	Yup (9 th pair of control points)
0	No. of straight dikes								
0	No. of right dikes								
0	No. of compound dikes × 2								

CCHE mesh generator can identify three kinds of hydraulic structures: *straight dikes*, *islands*, and *compound dikes*. “Straight” means that the dike is relatively straight and it

may or may not be attached to the bank. For the domain with some hydraulic structures, not only the physical coordinates but also the relative *location* of the structures must be known. In general, “Location” means the global number of the control points of the structures.

For each of the hydraulic structures, the format is different. Tab 3-1 only shows the format of the islands. The formats of *straight dikes* is shown in Tab 3-2.

Tab 3-2 Format of Right Dikes

No. of straight Dikes									
No.	Location	Xdn(1)	Ydn(1)	Xup(1)	Yup(1)	Xdn(2)	Ydn(2)	Xup(2)	Yup(2)
.....									
Note: 1. The straight dike must have only two pairs of control points, otherwise it will be treated as an island.									
2. “Location” here is the global number of the first pair of control points of the dike.									

The *compound dike* is defined as the mixture dike consisting of two dikes with different angles as shown in Fig 3-31.

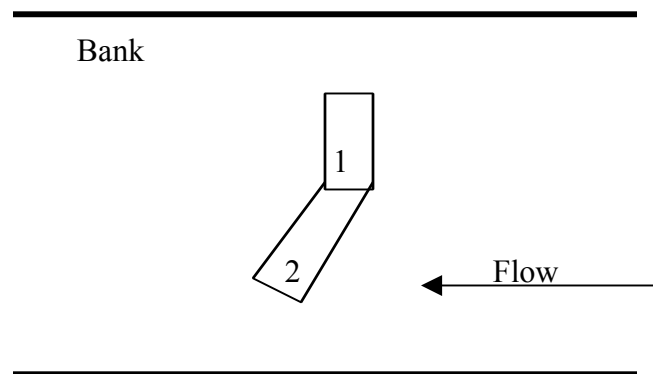


Fig 3-31 Compound Dikes

So there are three control points on each side of the dike and the river is divided into four sections in the transverse direction.

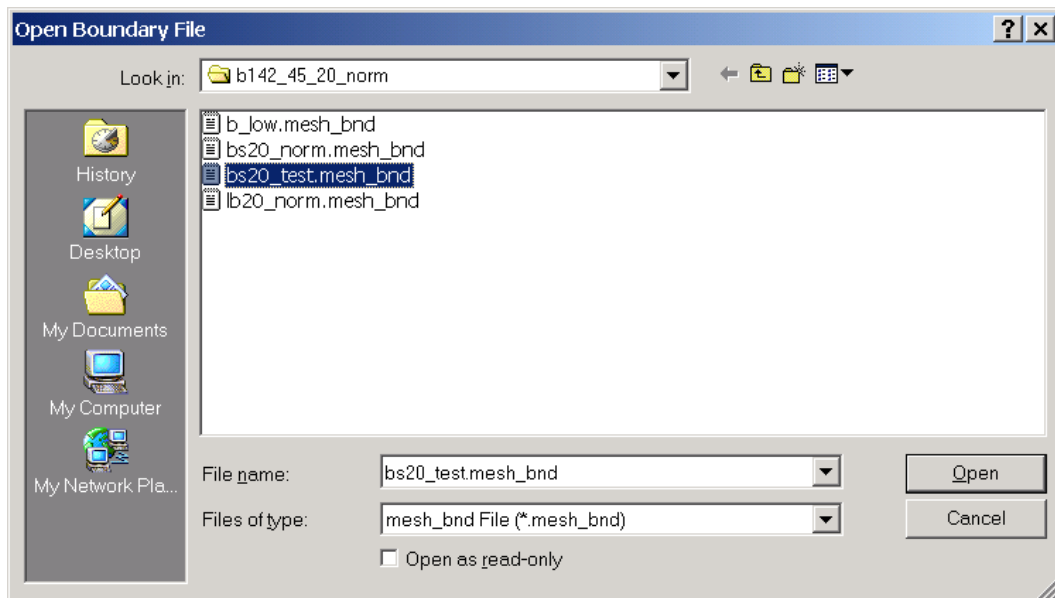
Tab 3-3 Format of Compound Dikes

No. of Compound Dikes $\times 2$							
<i>i.d of side i.d of four bnds of four sections</i>							
No.	Location	Xc(1)	Yc(1)	Xc(2)	Yc(2)	Xc(3)	Yc(3)
.....							
Note: 1. “-1” means left side of dike and “1” means right side. (<i>i.d of side</i>)							
2. “1” means wall bnd and “0” means no bnd. (<i>i.d of four bnds of four sections</i>)							

3.2.4 Generate Algebraic Mesh

Using the data of Tab 3-1 as boundary file, now the user can generate the mesh step by step.

Step-1: Input the boundary file. Click “File” menu and choose “Open Bnd File”.



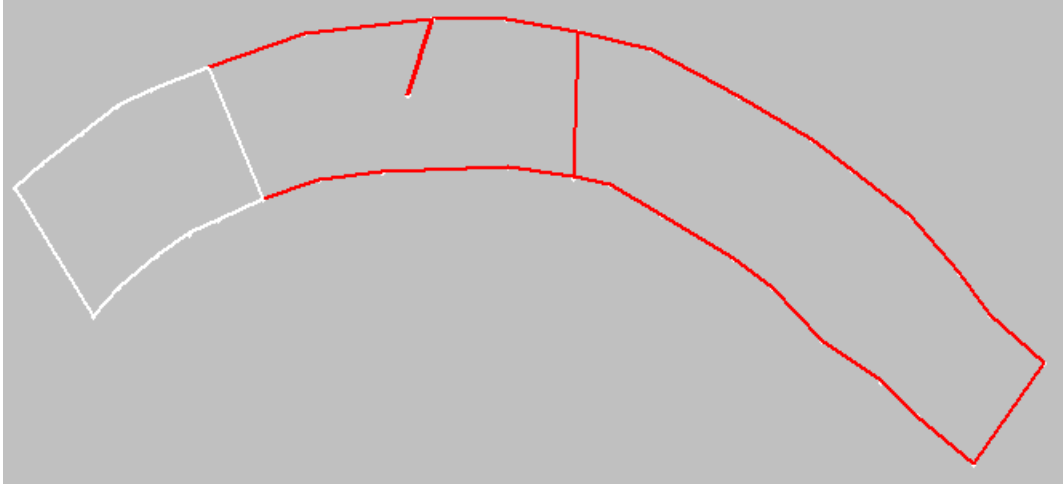


Fig 3-32 Open Boundary File

Step-2: Generate the mesh for block 2. Because block 2 has a dike, it is recommended to begin with this block. For the domain with a dike, the user always cares more about the area near the dike. On each side of dike and around the tip, the mesh should be denser. For this special consideration, stretching function is needed to generate the non-uniform mesh.

First, evaluate the mesh size: $I_{\max} \times J_{\max} = 93 \times 45$ and then generate the mesh using the default parameter values. However, the mesh is nearly uniform which is undesired (Fig 3-32)

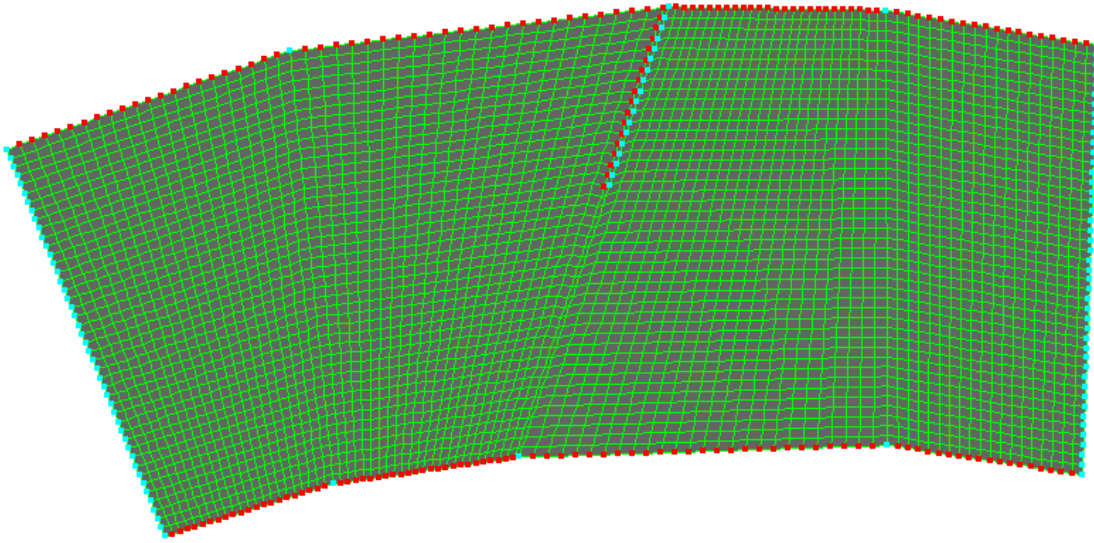
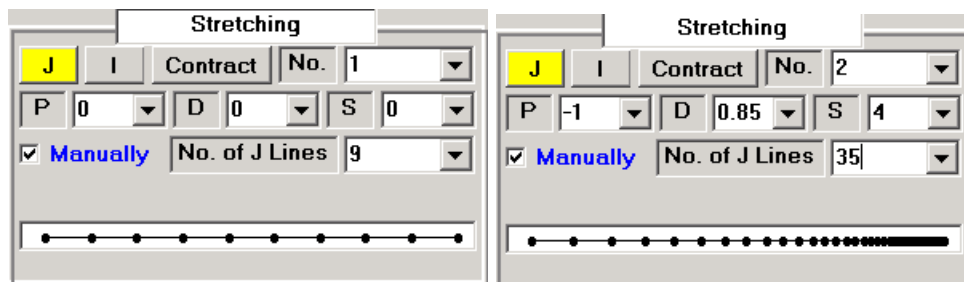


Fig 3-33 Uniform Mesh for Block 2

Now go to the “Stretch Function” group and adjust parameters. Block 2 has 6 pairs of control points which divide the block into 5 subsections. The user needs to adjust the stretching parameters for each subsection. Check “Manual Distribution” and adjust the parameters section by section (Tab 3-4).

Tab 3-4 Parameters of each subsections

	Sub-1	Sub-2	Sub-3	Sub-4	Sub-5
N.I.L	9	35	3	37	8
P	0	-1	0	-1	0
D	0	0.85	0	0.15	0
S	0	4.0	0	4.0	0



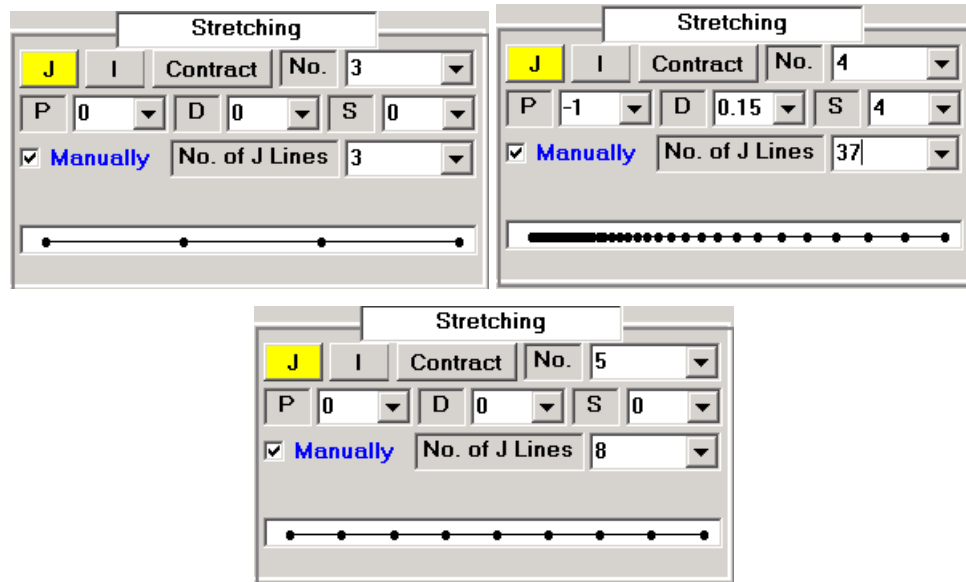


Fig 3-34 Parameters of each subsection

Now click “Algebraic” again and a non-uniform mesh is obtained (Fig 3-35).

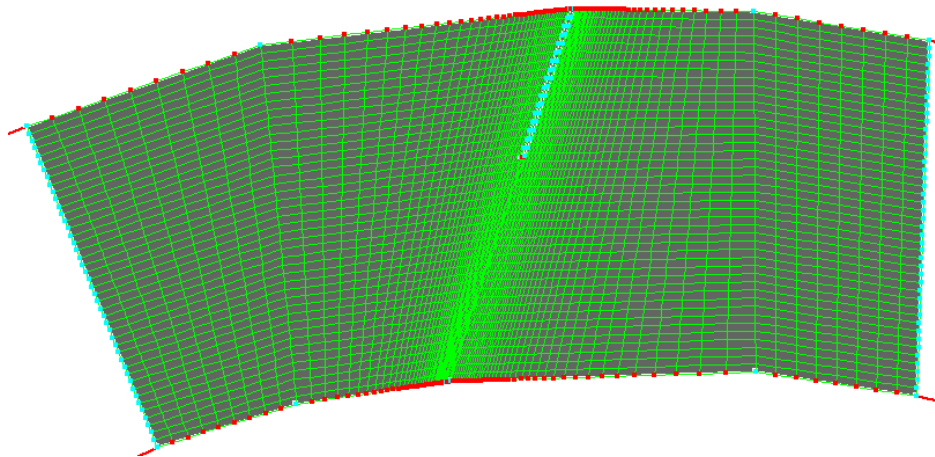


Fig 3-35 Non-uniform mesh of Block 2 (1)

However, the mesh lines are also required to be contracted to the tip in the transverse direction. Thus, the tip is considered as a contraction point, which divides the cross section into two parts. In each part, the stretching parameters are to be evaluated. “R.L.C.P” is defined as the ratio of the distance from the contraction point to the

downward boundary to the width of the cross section where the contraction point is located.

Tab 3-5 Parameters of each part

	N.J.L	R.L.C.P	P	D	S
Part-1	24	0.493	1.0	0.5	2.0
Part-2	22	1.0	1.0	0.75	3.0

Choose “Contraction” option and then input the parameters part by part (Fig 3-36).

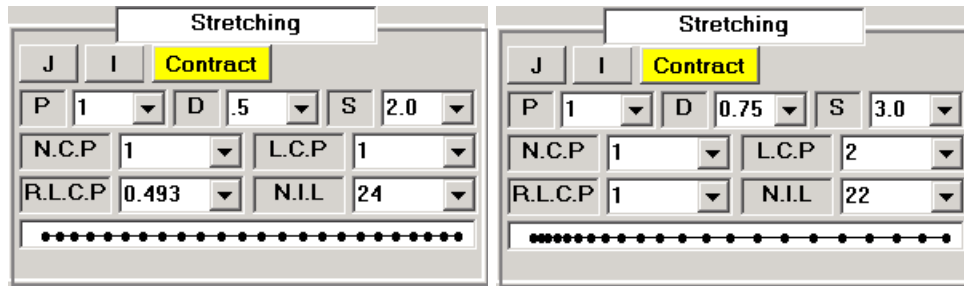


Fig 3-36 Parameters of Contraction

Click “Algebraic” again and another non-uniform mesh is obtained (Fig 3-37).

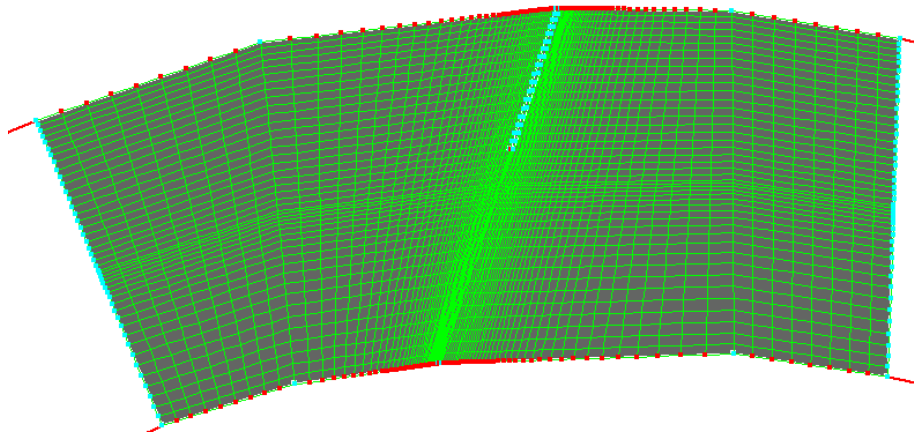


Fig 3-37 Non-uniform Mesh of Block 2 (2)

Step-3: Generate the mesh for block 1. Set the mesh size: $I_{\max} \times J_{\max} = 18 \times 45$ and then click “Algebraic” (Fig 3-38). Note that the contraction point set in block 2 are also

extended to both in block 1 and block 3. The automatic distribution will be used for each subsection in block 1, so make sure to uncheck the “Manual Distribution”, otherwise all other distribution settings in block 2 will be used in this block.

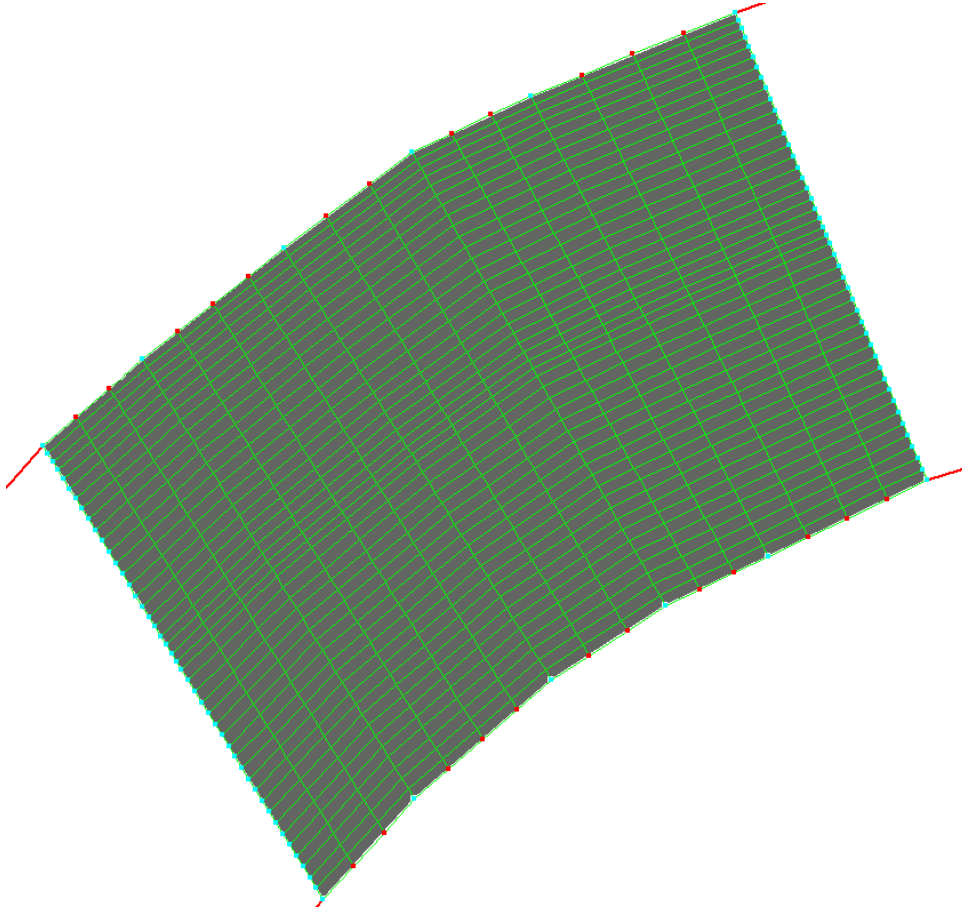


Fig 3-38 Mesh of Block 1

Step-4: Generate the mesh for block 3. Evaluate the mesh size: $I_{\max} \times J_{\max} = 33 \times 45$ and then click “Algebraic” (Fig 3-39).

Step-5: Connect the meshes of each block. Click “Connect” and a complete mesh is obtained. (Fig 3-40)

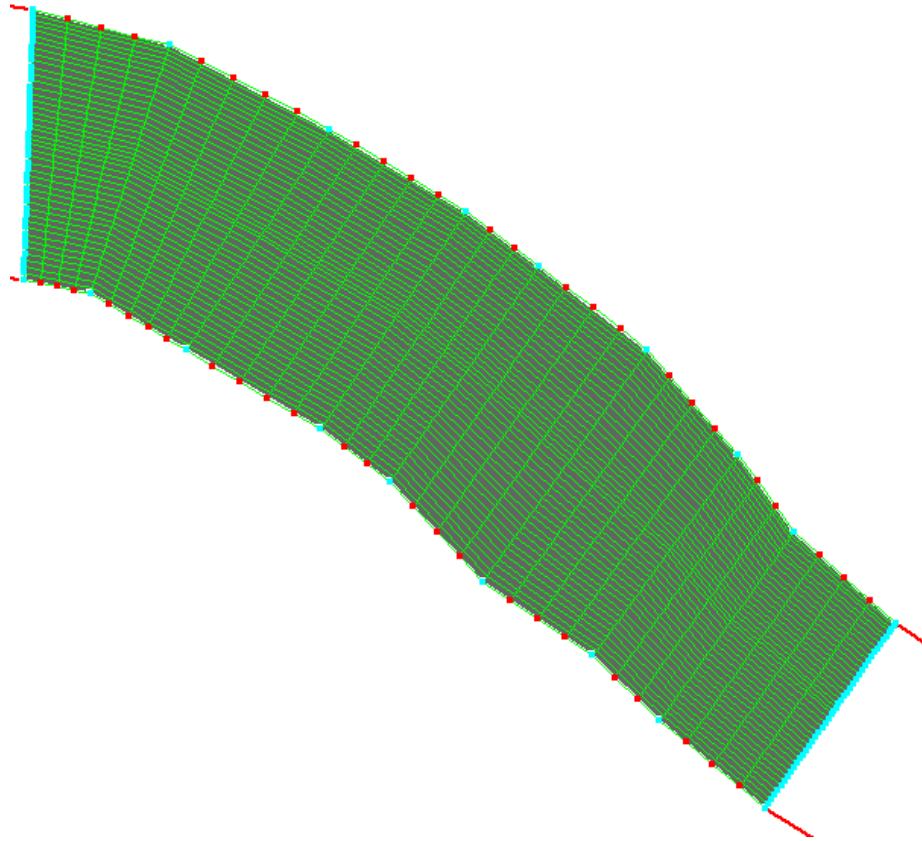


Fig 3-39 Mesh of Block 3

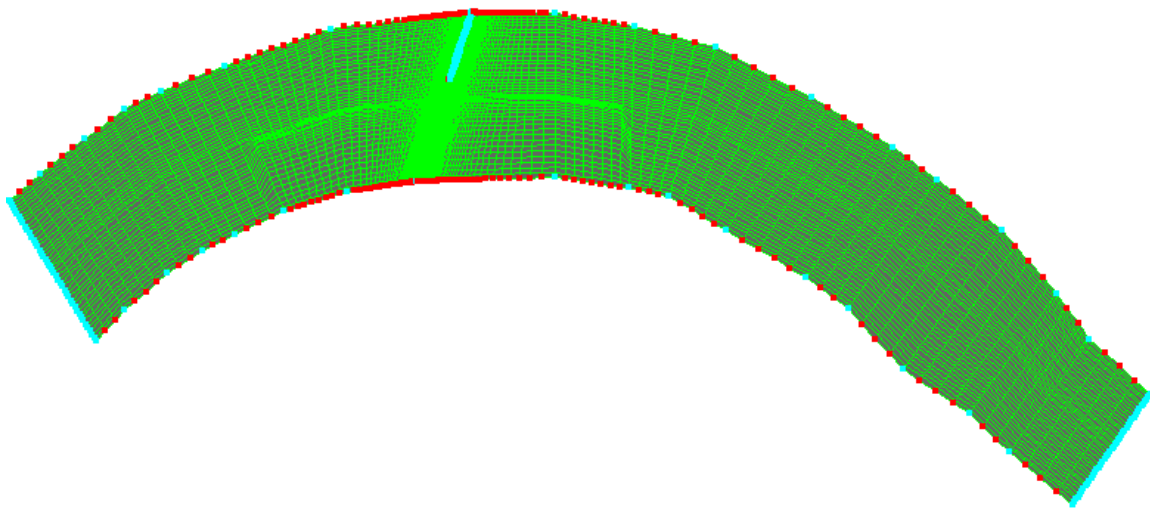


Fig 3-40 The Complete Mesh

Step-6: Interpolate the bed elevation. The file with the extension “.mesh_xyz” is identified as the bathymetry database file, which is used to interpolate the bed elevation. The format of this file is shown in Tab 3-6.

Tab 3-6 Format of “.mesh_xyz” file

No. of Total Points
X, Y, Z

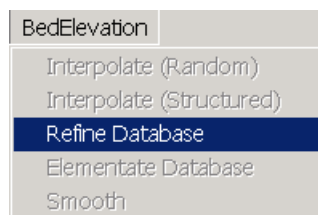
Currently, two interpolation methods are used in CCHE mesh generator. For non-structured survey data, the inverse-distance interpolation due to surrounding available survey points is applied. As for the structured survey data, such as those measured section by section along a channel, a refined-database method due to thalweg is applied.

The file with the extension “.mesh_mcs” is identified as the measured cross section data file, which is used to refine the original database. The format of this file is shown in Tab-3-7.

Tab 3-7 Format of “.mesh_mcs” file

No. of Total Cross Section
Index of Cross Section No. of Points on the Cross Section
X, Y, Z
.....(next cross section)

If the structured measured cross section data is available, the user can choose the refined-database method to improve the accuracy of the interpolation. Click “Topography” menu and choose “Refine Database”, the refinement process will begin. At the end of this process, the refined database will be saved.



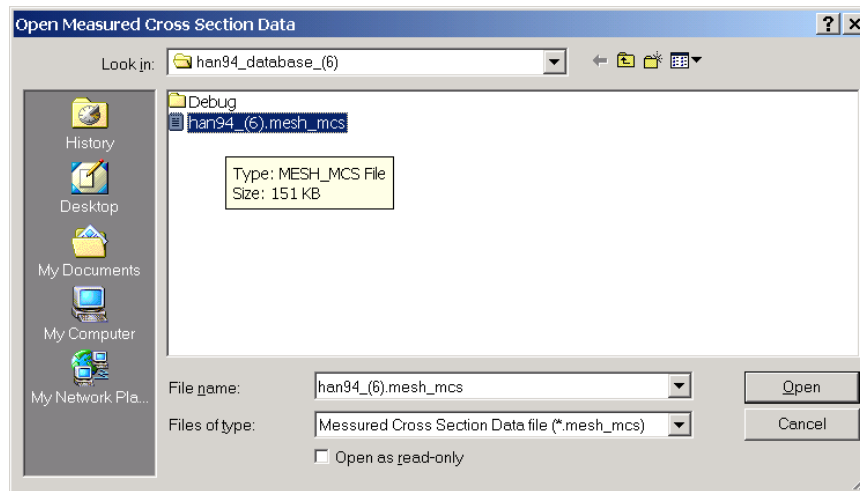


Fig 3-41 Open a “.mesh_mcs” file

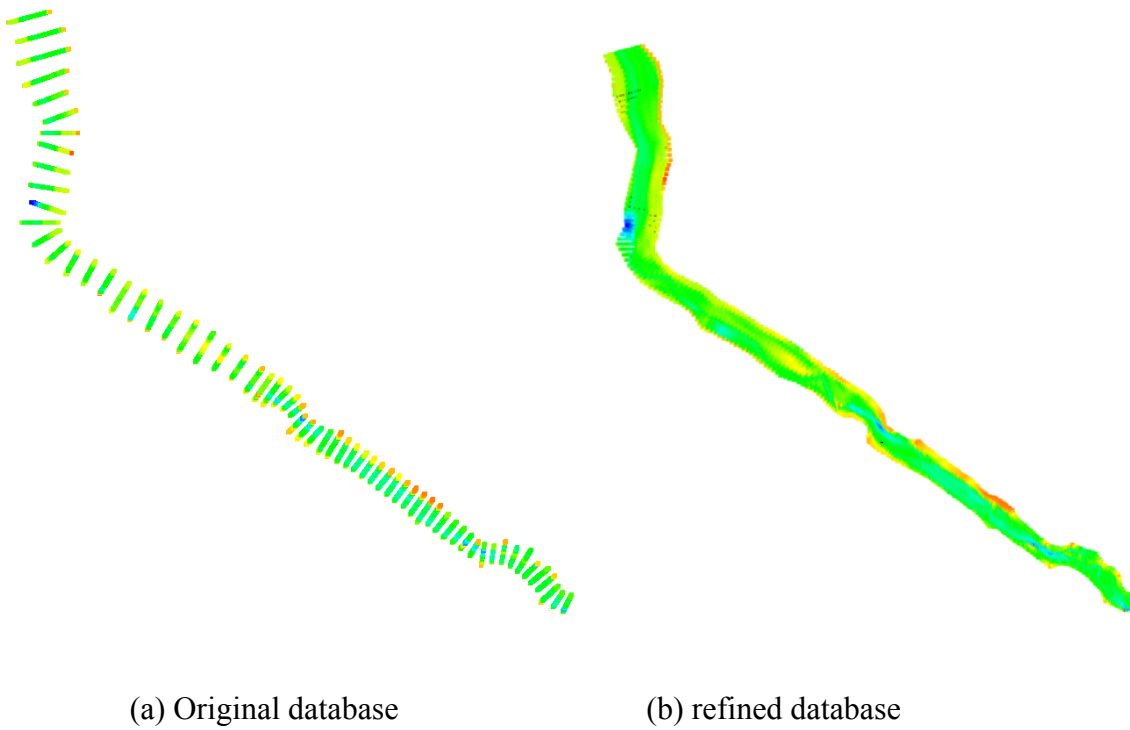


Fig 3-42 Original database and refined database

If the bed bathymetry database file is ready, click “Bed” and choose “Interpolate (Random)”, and then the interpolation begins.

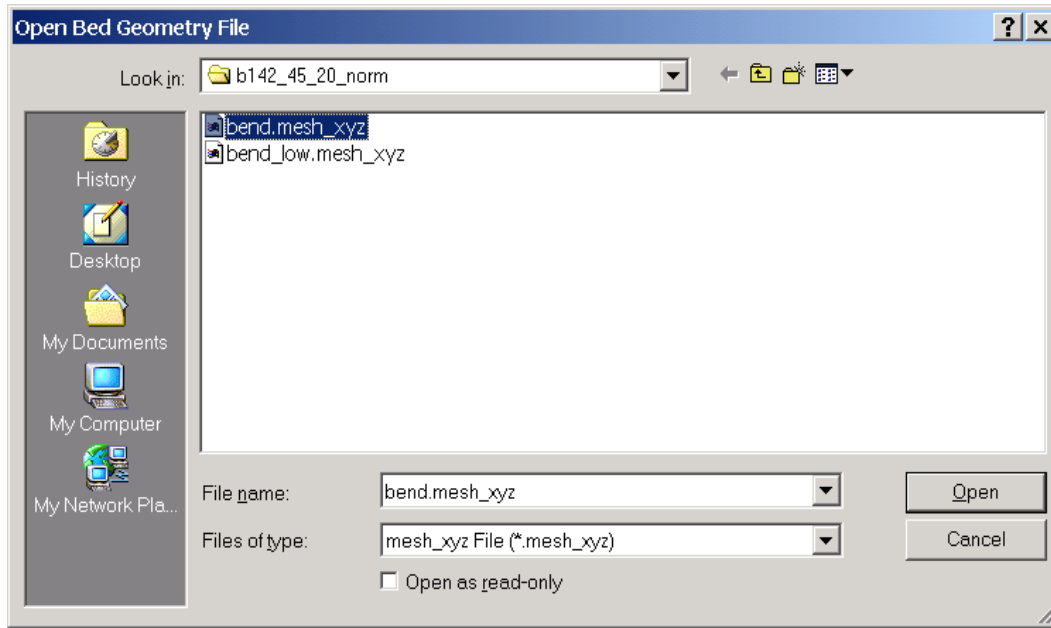


Fig 3-43 Open the Geometry File

Step-7: Save the mesh as .geo file. So far the algebraic mesh is completed. If the mesh is satisfactory, the user can save the mesh as .geo file, which is to be used by CCHE2D and CCHE3D model.

3.2.5 Generate Near-Orthogonal Adaptive Mesh

CCHE2D Mesh Generator solves the modified Poisson Equation to generate near-orthogonal adaptive mesh. The three non-negative parameters, orthogonality, smoothness, and adaptivity, need to be evaluated before computing. Among these three parameters, the orthogonality and smoothness, whose sum is one, are dependent upon each other and they are both less than one; and the adaptivity is independent parameter. Note: if the option “Keep the measured cross section” is checked, those sections will be fixed during the generation. This option is available for both the numerical methods and the smoothing methods.

3.2.6 Generate Variational Adaptive Mesh

CCHE2D mesh generator solves the Variational Laplace Equation to obtain the variational adaptive mesh. Similarly, the three non-negative parameters, orthogonality, smoothness, and adaptivity, need to be evaluated before computing. However, in this approach, these three parameters are all independent upon each other.

3.2.7 Generate Smooth Mesh

There are two methods available to generate smooth mesh: one is through solving Laplace Equation for conformal mapping and the other is through the Laplacian smoothing scheme.

Check the option “Local Smoothing”, the user can specify the smoothing area by choosing two mesh nodes which are the diagonal points of the quadrilateral area specified for local smoothing, otherwise the whole domain will be smoothed.

3.3 How to Use Stretching Function

The stretching function is often used to distribute nodes along the boundaries. In CCHE2D mesh generator, the stretching function can be applied in two directions: J and I. Note that at each time the stretching function can be applied to only one line or subsection. Here an example is presented to illustrate how to use the stretching function to control the mesh density. As shown in Fig 3.43, the mesh size in a rectangular domain is 35×35 and it is required that: (1) repulsion occurs at the node A, B, C, and D; (2) contraction occurs at the line EF and GH.

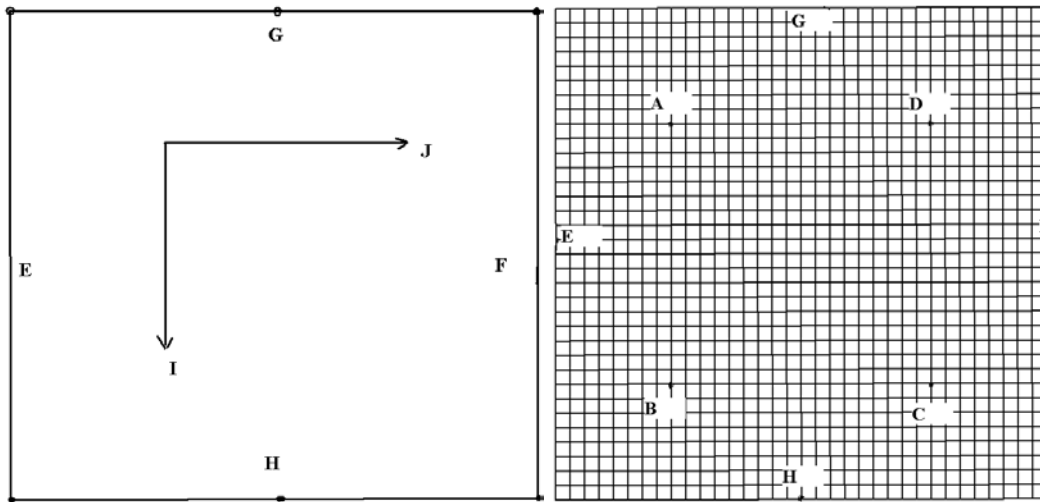
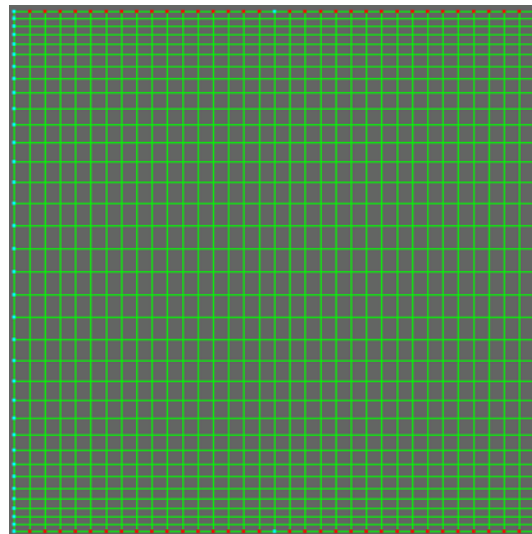
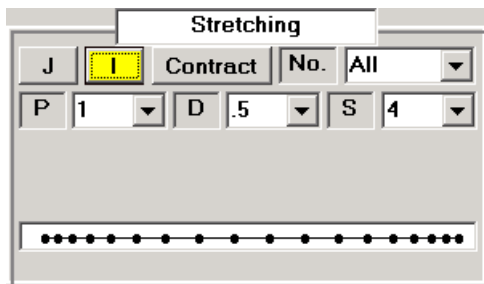


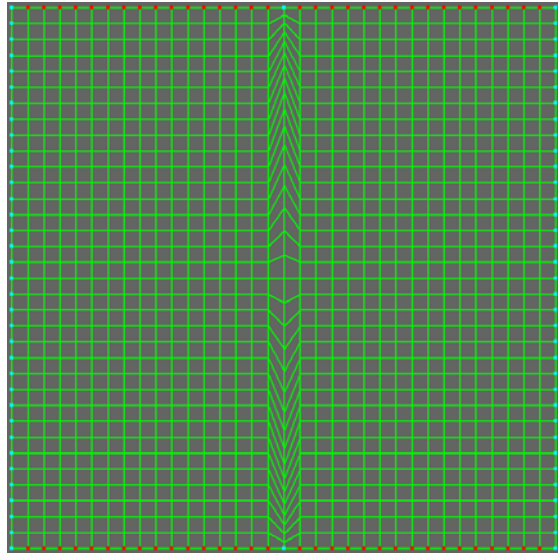
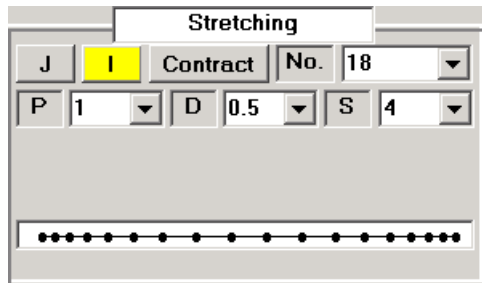
Fig.3.44 rectangular domain

3.3.1 Apply Stretching Function in I Direction

In I direction, in default there is only one sub-section. As shown in Fig 3.44, one can apply the stretching function for all I lines or just one specific I line by choosing option “I Line”.



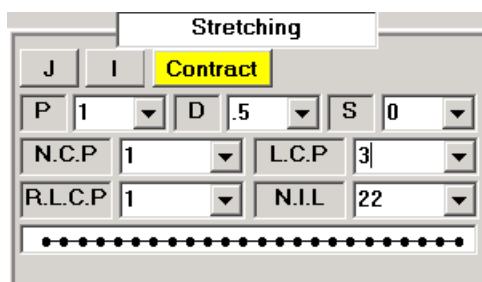
(a) for all I lines



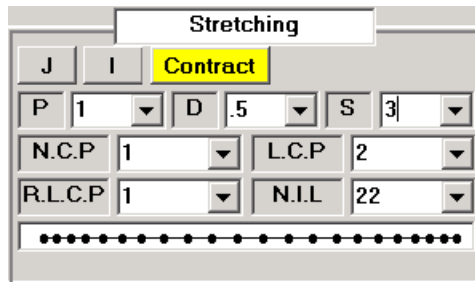
(b) for No.18 I line

Fig 3.45

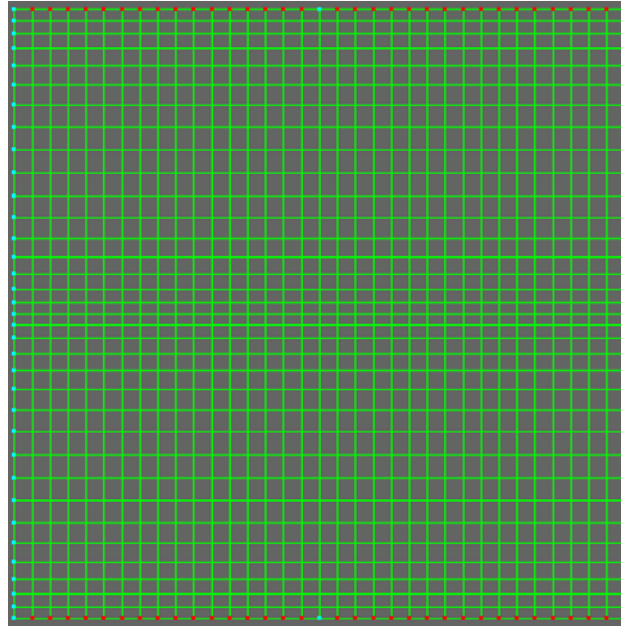
In this example, because it's required the contractions at line EF, one needs to add contraction point in this direction. The contraction points also divide I line into several sub-sections where the stretching function is used. "N.C.P" represents the number of contraction points added in I direction. "L.C.P" represents the labels of sub-sections created by the contraction points. "R.L.C.P" represents the relative location of the contraction points and it is in the range of 0 and 1, that is, $0 \leq R.L.C.P \leq 1$. "N.I.L" represents the number of I lines in the selected sub-section. As shown in Fig 3.45, first click option "I Line" and option "Contraction" in group of stretching function, and then evaluate the parameters in pink color.



(a) first sub-section



(b) second sub-section

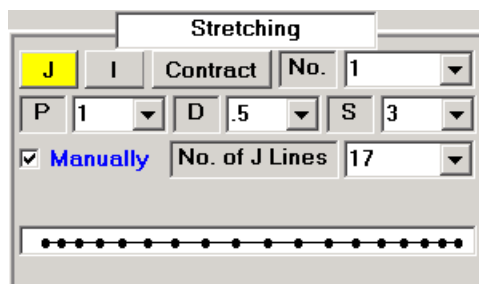


(c) mesh

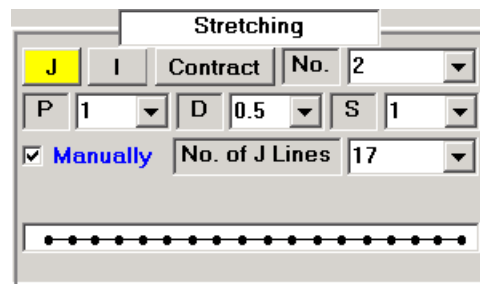
Fig 3.46 Application of stretching function in I direction

3.3.2 Apply Stretching Function in J Direction

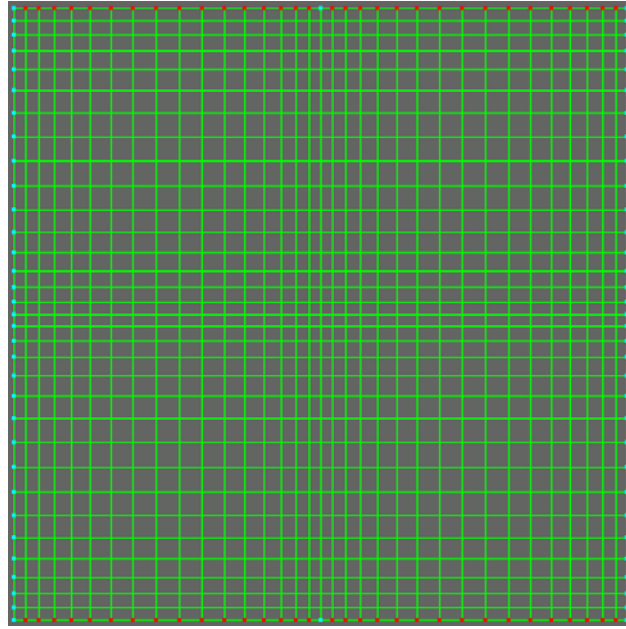
Secondly, the stretching function is applied in J direction. As shown in Fig 3.43, in J direction there are three control points which divide the boundary into two sub-sections. The stretching function is applied to these two sub-sections respectively. As shown in Fig 3.46, first click option “Bnd” and check “Manual Distribution” in group of stretching function, and then evaluate the parameters in blue color. “N.J.L” represents the number of J lines specified in the selected sub-section.



(a) first sub-section



(b) second sub-section



(c) mesh

Fig 3.47 Application of stretching function in J direction

3.4 How to Edit Mesh

3.4.1 Edit Boundary

When a boundary file or a work-space file or a Geo file is opened, the user can edit the boundary if necessary.

This can be done by the following procedure: (1) Click “Boundary” button of “Edit” group to activate this function. You will find that both the button “Boundary” and “Move Node” become yellow. (2) Click the control point you want to move and hold it. Move it to the desired place and then release it. (3) Repeat step (2) and you can edit more control points. (4) Click “Save as Bnd File” of menu “File” and all the changes will be saved into a Bnd file.

3.4.2 Adjust Mesh Nodes’ or Lines’ Location

When the mesh is completed, the “Edit” functions are available for adjusting local nodal distribution.

- Click “Move Node” and then click the node to be moved. Hold the left button of the mouse and move the node to the desired location. Note that only the internal nodes can be moved. If you want to move the boundary nodes, click “boundary” and then you can move both the internal nodes and the boundary nodes.
- Click “Add I Line” or “Add J Line” and then click the location where one *I* line or *J* line is to be added, one *I* line or *J* line will be added into the mesh. Note: make sure the location clicked is within the mesh, otherwise nothing will happen.
- Click “Delete I Line” or “Delete J Line” and then choose the *I* line or the *J* line is to be deleted, click it and then this line will be excluded.
- Click “Undo” and the changes of the previous step will be erased.
- Click “Save Change”, all the changes made since the last “save change” will be saved. Note that the changes are stored only into the memory, not the file.

3.4.3 Modify the Properties of Mesh Nodes

Sometimes the user may want to evaluate or modify the properties of the mesh nodes, which are (X, Y, Z) coordinates, nodal type (I.D) and Manning’s n . Click “Edit” button in “Mesh Information” group, a edit-pen window will pop out.

There are two pages of this window and the user can push “>>” or “<<” button to switch between them.

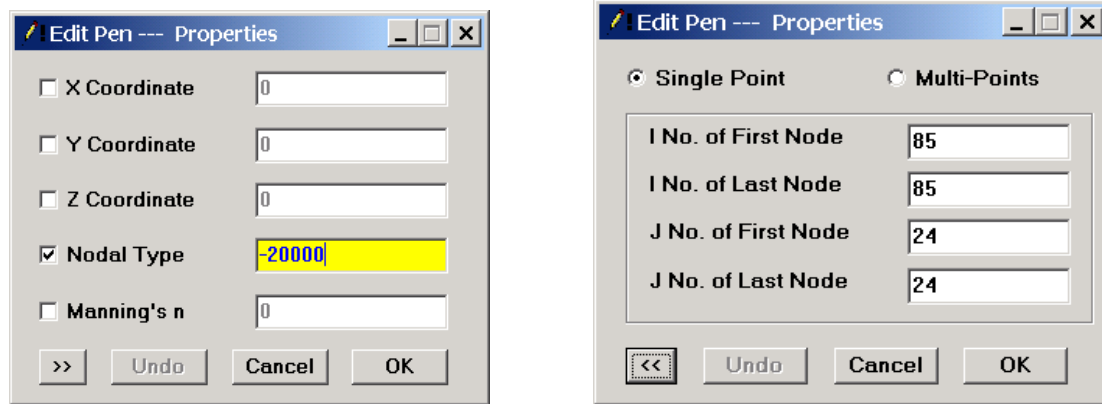


Fig 3-48 Edit-pen window

In the first page, there are five check boxes and five inputting boxes, which represent the five properties of the mesh nodes. Check one box and the corresponding inputting box will turn yellow. Click “OK” and the values of the checked inputting boxes will be stored into the edit pen. Click “Cancel” and the values stored in the edit pen will be cleared.

In the second page, there are two options and four inputting boxes. The user can either choose “Single Point” to evaluate or modify the properties of one specified mesh node using the values stored in the edit pen or choose “Multiple Points” to evaluate or modify the properties of the multiple specified mesh nodes.

So there are two ways to modify the properties of mesh nodes. The default way is point by point. When using this way, the user doesn’t need to go to the second page. After clicking “Ok”, the user can modify the properties of the mesh node through the following operations: Press “Ctrl” and hold it, then use right button of the mouse to click the desired node.

If the user chooses “Multiple Points”, he can use the four inputting boxes to specify the range of the mesh nodes where the properties will be set or modified. Click “OK” and then the properties of the mesh nodes in this specified range are changed.

The user can use “Undo” to recover the changes of the previous step before “Save Change”.

3.5 View Mesh in Different Ways

After the interpolation of the bed elevation is finished, the user can view the mesh in different ways by pushing the buttons in the “Display” groups.

3.5.1 View Bed Topography

Click “Bed” and the color-shade picture of the bed topography will be shown. (Fig 3-49)

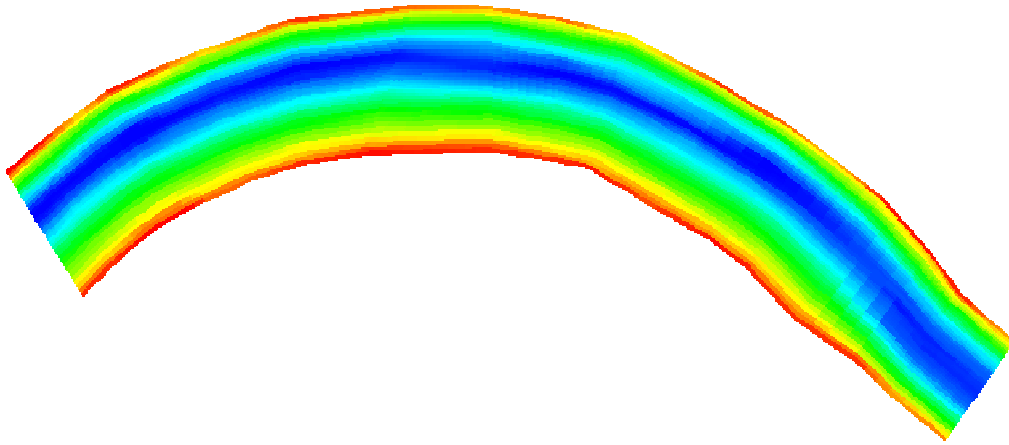


Fig 3-49 Bed

3.5.2 View Bed Topography with Mesh on

Click “Grided Bed” and the color-shading picture of the bed topography with grid will be shown. (Fig 3-50)

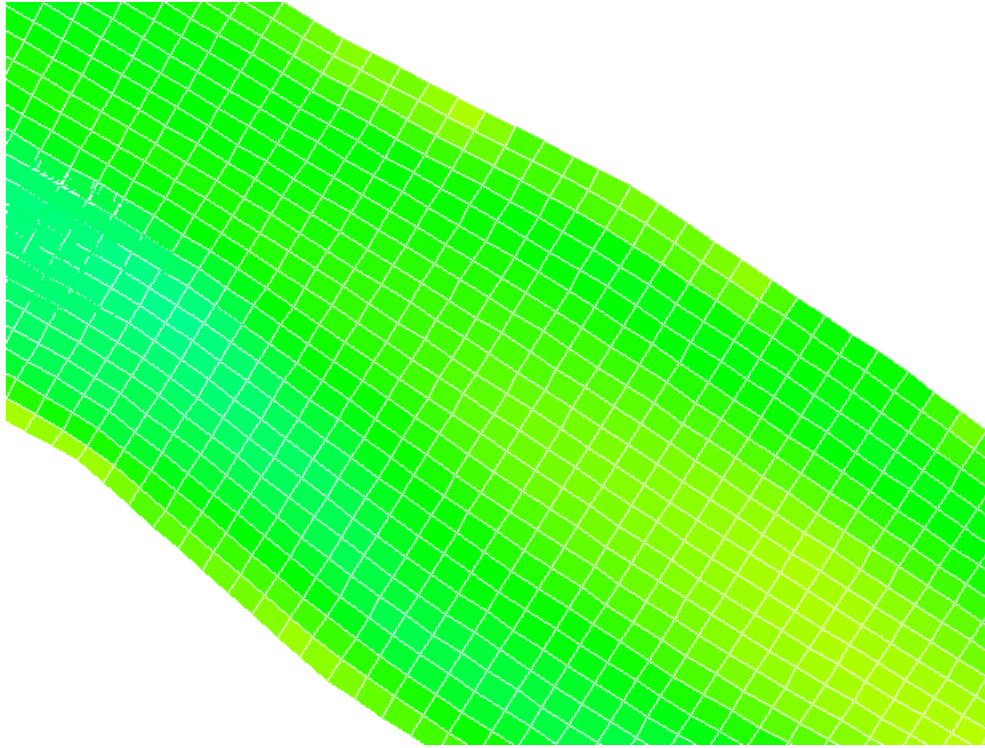
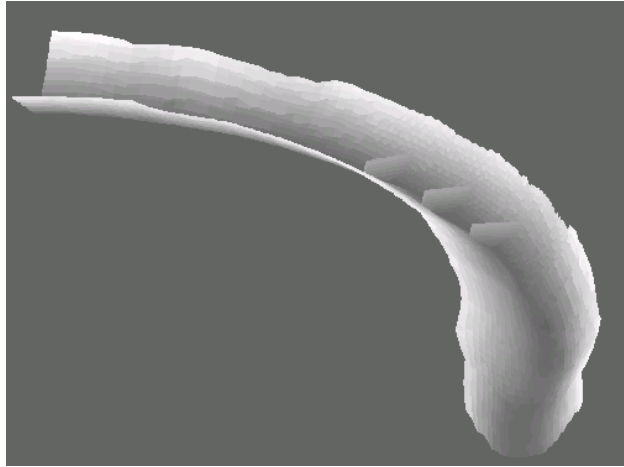


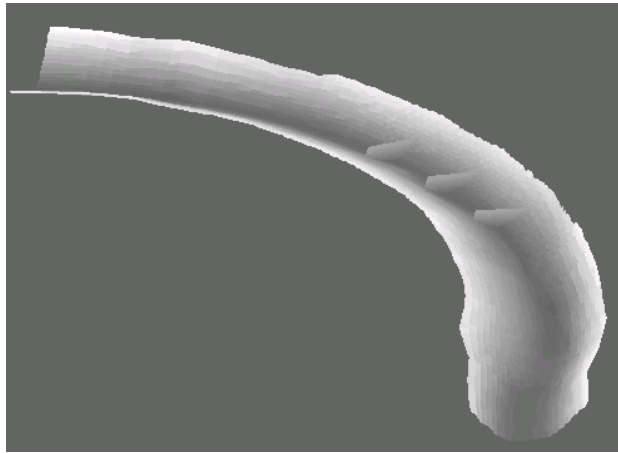
Fig 3-50 Bed with Grid

3.5.3 View 3D Mesh

Click “3D” for the odd time and the 3D topography with mesh will be shown; click it for the even time and the 3D topography without mesh will be shown. The user can rotate the 3D topography by moving the three scroll bars, “X”, “Y” and “Z”. The user can also increase the scale in Z direction by clicking “ $\leftarrow \rightarrow$ ” button and decrease it by clicking “ $\rightarrow \leftarrow$ ” button.



(a) 3D Topography by increasing Z scale



(b) 3D Topography by decreasing Z scale

Fig.3-51 3D Topography

3.5.4 View Mesh Nodes

Click “Node” and the mesh nodes colored due to bed elevation will be shown. (Fig 3-52)

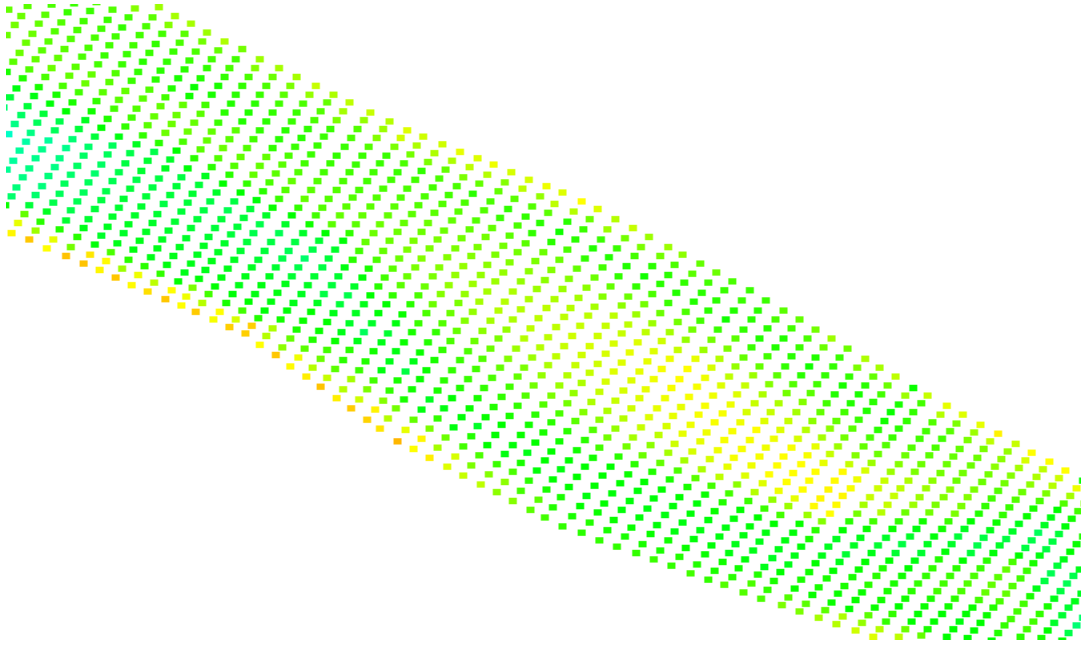


Fig 3-52 Mesh Nodes

3.5.5 View Cross Section

The user can view the cross sections through the cross section sub-window.

- Click “I” option and the transverse cross section will be shown while click “J” option and the longitudinal cross section will be shown. The user can move the scroll and change the label of the cross section to be shown.
- Check “Multi” and up to five cross sections will be shown at the same time. “Multi. No.” means the number of the sections the user wants to show at the same time. Five colors have already been assigned to these five cross sections. Of course, the user can specify these five cross sections by changing the labels of them.
- The user can also choose different types of “X-axis”. Two types, “Cross Section No.” and “Distance”, are available.

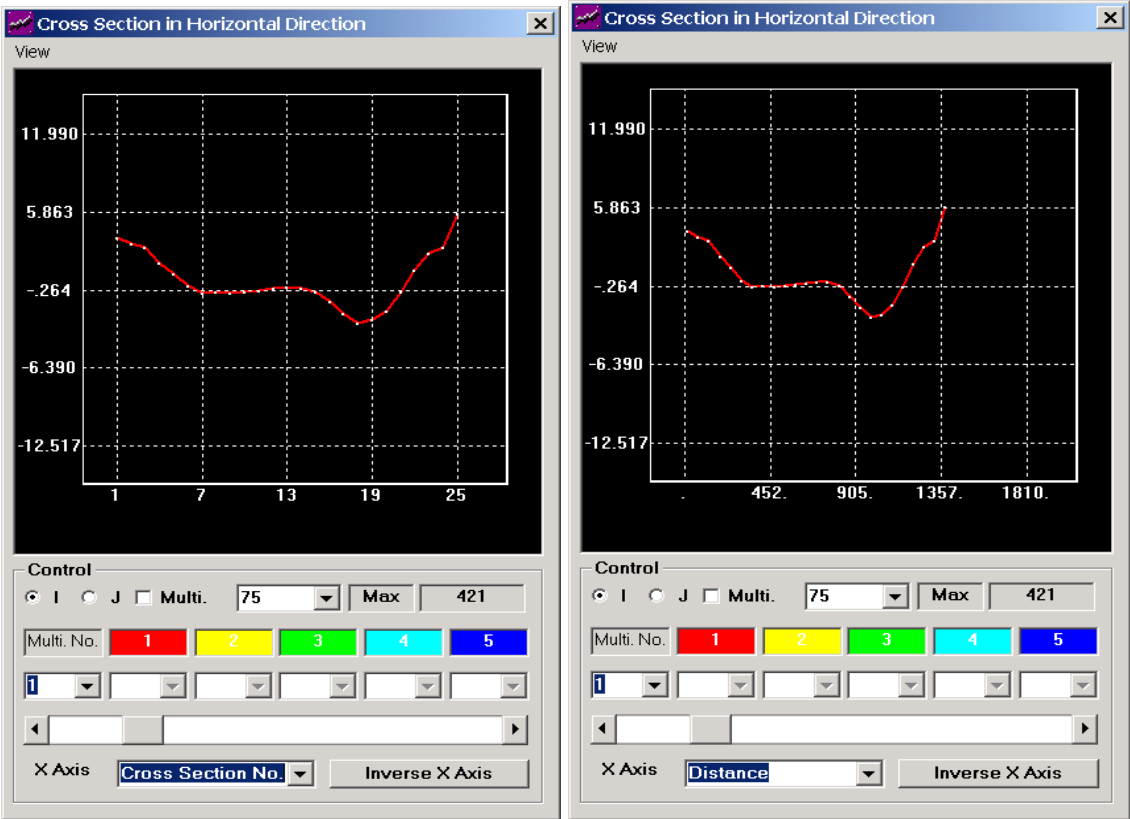


Fig 3-53 Different types of X-axis

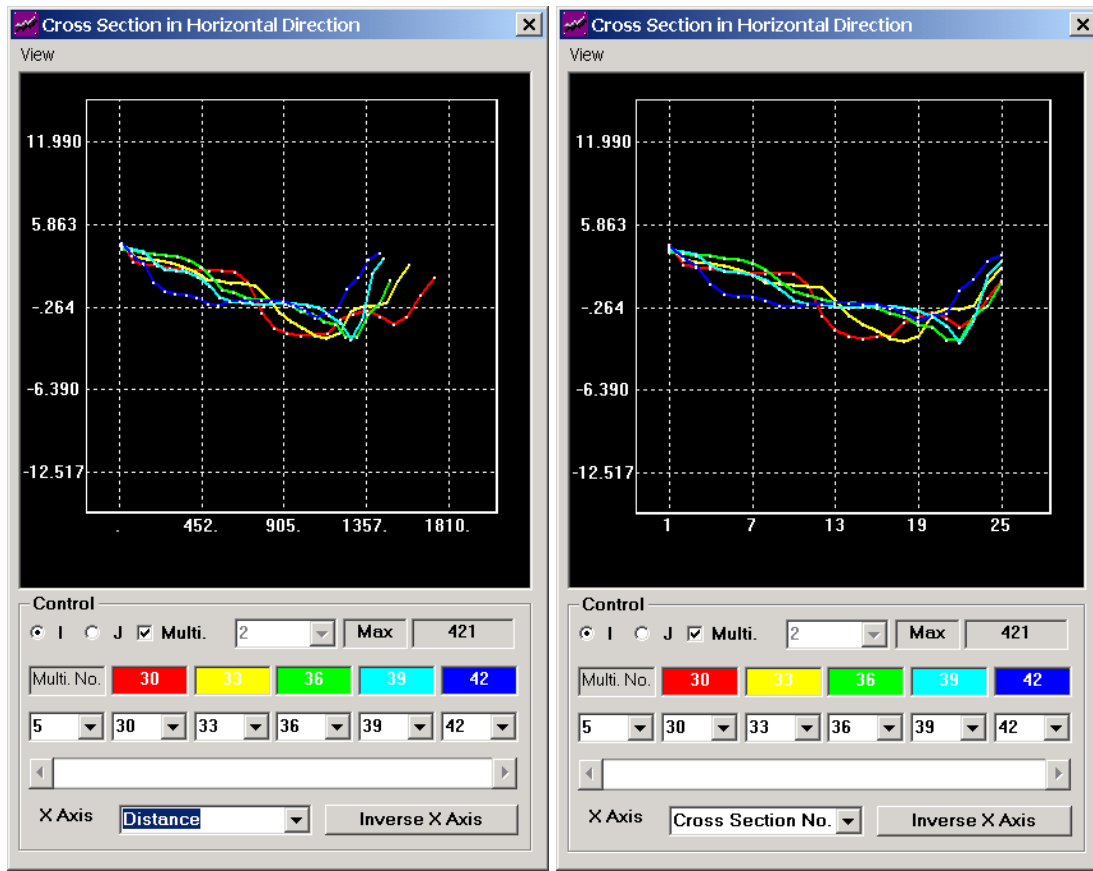


Fig 3-54 Multiple Cross Sections

3.5.6 Import Associated Topography

When a boundary file or a work-space file or a Geo file is opened and if the corresponding topography database is available, the user can import the database as background.

The process is composed of two steps: (1) Click “Load Topographic Database” and a question ---“Do you want to import the associated geometry?” will be asked. Choose “Yes”. (2) Open the associated database file.

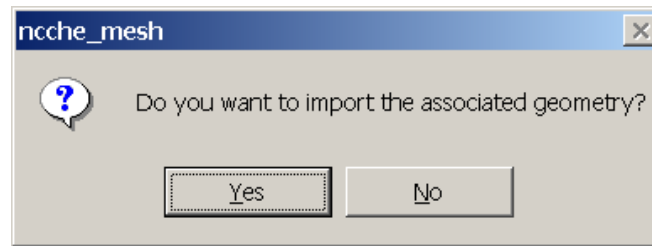


Fig 3-55 Import the associated geometry

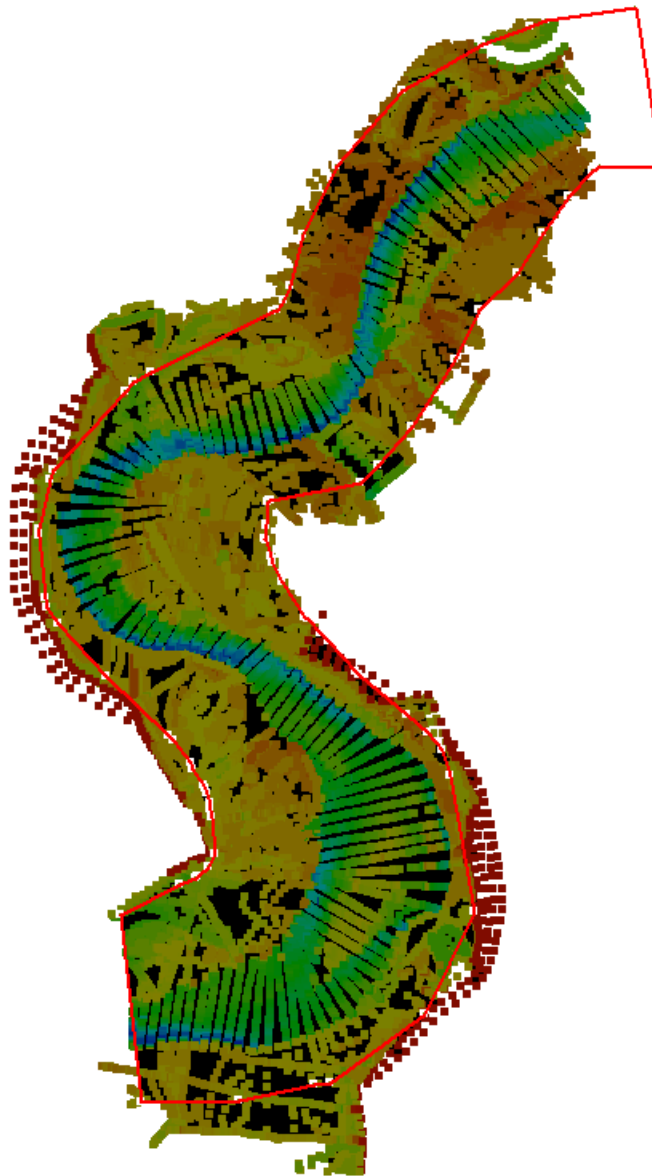


Fig 3-56 Add a Background

3.6 System Requirements

The current version of CCHE2D mesh generator is developed for Microsoft Windows 98, Windows NT, Windows 2000 and Windows XP. The users of other operating system should be careful that the mesh generator could not work or work incorrectly in those systems. The resolution of 800×600 is the minimum requirement for CCHE2D mesh generator and it's recommended to use higher resolution. And the best resolution is 1280×1024 .

3.7 Review of Formats of Files Used in CCHE Mesh Generator

There are five files used in CCHE mesh generator: the mesh boundary file (*.mesh_bnd), the bathymetry database file (*.mesh_xyz), the mesh work-space file (*.mesh_wsp), the measured cross section file (*.mesh_mcs), and the output mesh file for CCHE2D and CCHE3D models (*.geo).

The user needs to be familiar with the formats of these files except the mesh work-space file. The formats of these files in Fortran are reviewed in detail as follows for convenient reference.

3.7.1 The Mesh Boundary File (*.mesh_bnd)

The format of mesh_bnd file is a little complex. The reading format of this file in Fortran code is as follows.

```
Open(11, file= "test.mesh_bnd")  
//---input global information  
Read(11,*)mb_imax, mb_jmax, mb_npmax
```

```
Read(11,*)mb_nrdike, mb_nisland, mb_ncdike
//---input data block by block
DO i = 1, mb_imax
  DO j = 1, mb_jmax
    k = (i-1)*mb_jmax+j
    //---input control points of the block k
    Read(11,*)mb_nc(k)
    Read(11,*)mb_bnd(k,1), mb_bnd(k,2), mb_bnd(k,3), mb_bnd(k,4)
    DO n = 1, mb_nc(k)
      Read(11,*)n, xcb(k,n), ycb(k,n), xct(k,n), yct(k,n)
    ENDDO
    //---input right-dike
    Read(11,*)mb_nrd(k)
    DO n = 1, INT(mb_nrd(k)/2)
      Read(11,*)n, n2, xdf1,ydf1, xdl1,ydl1, xdf2,ydf2,xdl2,ydl2
      Lid_rd(k, 2*n-1)=n2
      Lid_rd(k, 2*n)=n2+1
      xdf(k, 2 * n - 1) = xdf1
      ydf(k, 2 * n - 1) = ydf1
      xdl(k, 2 * n - 1) = xdl1
      ydl(k, 2 * n - 1) = ydl1
      xdf(k, 2 * n) = xdf2
      ydf(k, 2 * n) = ydf2
      xdl(k, 2 * n) = xdl2
      ydl(k, 2 * n) = ydl2
    ENDDO
    //---input island
    Read(11,*)mb_nh(k)
    DO n = 1, mb_nh(k)
      Read(11,*)mb_nhc(k,n)
      DO m = 1, mb_nhc(k,n)
```

```

                                Read(11,*)ii, lid_nh(k,n,m), xhb(k,n,m),yhb(k,n,m),    &
                                Xht(k,n,m), yht(k,n,m)
                                ENDDO
                                ENDDO
                                //---input compound dike
                                Read(11,*)mb_ncd(k)
                                DO n = 1, mb_ncd(k)
                                    Read(11,*)id_cd(k,n), (id_bnd(k,n,jj), jj = 1, 4)
                                    Read(11,*)lid_cd(k,n), (xcd(k,n,jj), ycd(k,n,jj), jj = 1, 3)
                                ENDDO
                                ENDDO
                                ENDDO

```

The variables in the above Fortran code are:

//---global information

mb_imax = maximum number of blocks in i direction;
mb_jmax = maximum number of blocks in j direction;
mb_npmax = maximum number of control points for all blocks;
mb_nrdike = maximum number of right dikes;
mb_nisland = maximum number of islands;
mb_ncdike = maximum number of compound dikes;

//---control points of block--k

mb_nc(k) = number of control points of block—k;
mb_bnd(k, 4) = boundary type of the four faces of block—k;
xcb(k,n) = x-coordinate of n-th control point on the bottom boundary of block—k;
ycb(k,n) = y-coordinate of n-th control point on the bottom boundary of block—k;
xct(k,n) = x-coordinate of n-th control point on the top boundary of block—k;
yct(k,n) = y-coordinate of n-th control point on the top boundary of block—k;

//---right dikes (two-lined dike)

mb_nrd(k)= number of right dikes×2 in block—k;
lid_rd(k, n) = relative location of the right dike n in block—k;

$xdf(k, n)$ = x-coordinate of the first control point of the right dike n in block— k ;
 $ydf(k, n)$ = y-coordinate of the first control point of the right dike n in block— k ;
 $xdl(k, n)$ = x-coordinate of the last control point of the right dike n in block— k ;
 $ydl(k, n)$ = y-coordinate of the last control point of the right dike n in block— k ;
//---islands
 $mb_nh(k)$ = number of islands in block— k ;
 $mb_nhc(k, n)$ = number of control points of island n in block— k ;
 $lid_nh(k, n, m)$ = relative location of m -th pair of control points of island n in block— k ;
 $xhb(k, n, m)$ = x-coordinate of m -th control point on bottom boundary of island n in
 block— k ;
 $yhb(k, n, m)$ = y-coordinate of m -th control point on bottom boundary of island n in
 block— k ;
 $xht(k, n, m)$ = x-coordinate of m -th control point on top boundary of island n in block— k ;
 $yht(k, n, m)$ = y-coordinate of m -th control point on top boundary of island n in block— k ;
//---compound dike
 $mb_cd(k)$ = number of compound dikes $\times 2$ in block— k ;
 $id_cd(k, n)$ = type of compound dike n in block k ; (-1, left side; 1, right side)
 $id_bnd(k, n, 4)$ = boundary type of four sections of compound dike n in block k ;
 $lid_cd(k, n)$ = relative location of compound dike n in block k ;
 $xcd(k, n, 3)$ = x-coordinate of three control points of compound dike n in block k ;
 $ycd(k, n, 3)$ = y-coordinate of three control points of compound dike n in block k ;

3.7.2 The Geometry 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

```

3.7.3 The Measured Cross Section 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)  ///---label and number of points of cross section k
      DO j = 1, np(k)
        ///---(x,y,z) of each points
        Read(13,*) xcs(k,j), ycs(k,j), zcs(k,j)
      ENDDO
    ENDDO

```

3.7.4 The 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

```

Conclusions

The CCHE2D Mesh Generator and its GUI have been developed for generating algebraic and numerical computational structured meshes.

It provides quite a few effective methods to generate mesh with its quality guaranteed. A flexible and powerful two-direction stretching function is applied to generate algebraic mesh; and both a Poisson-equation-based solver whose source terms are based on control functions and the variational approach with several optimal objectives controls are used to generate numerical mesh. The Laplacian smoothing scheme is applied to smooth mesh. A multi-blocks idea is applied to generate mesh for the domains with complicated geometry. A GUI is also developed to make the mesh generation more efficiently and easily.

During the process of its development, this generator has been applied to generate meshes for several computational simulation projects, such as the channel downstream of Wan'An Reservoir with many dikes, and an experimental channel with submerged weirs, etc. The experiences and the simulation results of CCHE2D and CCHE3D models have shown that CCHE mesh generator is capable of providing desired meshes for 2D and 3D modeling with high efficiency.

This is the version 2.0 of CCHE2D mesh generator and it will be improved by adapting new techniques.

References

1. Akcelik, V., Jaramaz, B., and Ghattas, O. (2001), "Nearly Orthogonal Two-Dimensional Grid Generation with Aspect Ratio Control", *J. Comput. Phys.* 171, pp. 805-821.
 2. Ascoli, E.P, Dandy, D.S, and Leal, L.G. (1987), "On Distortion Function for the strong Constrain Method of Numerically Generating Orthogonal Coordinate Grids", *J. Comput. Phys.*, v72, 513.
 3. Duraiswami, R. and Prosperetti, A. (1992), "Orthogonal Mapping in Two Dimensions", *J. Comput. Phys.*, v98, 254.
 4. Eça, L (1996), "2D Orthogonal Grid Generation with Boundary Point Distribution Control", *J. Comput. Phys.*, v125, 440.
 5. Ferziger, Jeol H. and Peric, Milovan (1997), "Computational Methods for Fluid Dynamics", Springer, New York.
 6. Fletcher, C.A.J. (1992), "Computational Techniques for Fluid Dynamics", Springer.
 7. Kang, I.S, and Leal, L.G (1992), "Orthogonal Grid Generation in a 2D Domain via the Boundary Integral Technique", *J. Comput. Phys.*, v82, 269.
 8. Sørli, Karstein (1999), "Boundary Control in Elliptic grid generation".
 9. Thomas, P.D., and Middlecoff, J.F (1980), "Direct Control of the Grid Point Distribution in Meshes Generated by Elliptic Systems", *AIAA* v18, No.6, pp. 652-657.
 10. Thompson, J.F, and Johnson, B.H (1985), "Development of an Adaptive Boundary-fitted Coordinate Code for Use in Coastal and Estuarine Areas", Miscellaneous paper HL-85-5, US Army Engineer Waterways Experiment Station, Vicksburg, MS.
 11. Thompson, J.F, Warsi, Z.U.A., and Mastin, C. Wayne (1985), "Numerical Grid Generation: Foundation and Application", North-Holland, New York.
 12. Thompson, J.F, Soni, B.K, Weatherill and N.P, Nigel (1998), "Handbook of Grid Generation", CRC press, New York.
-

13. Thompson, J.F (1996), “A Reflection on Grid Generation in the 90s: Trends, Needs, and Influences”, 5th International Conference on Grid Generation in CFS, Mississippi State University.