# RECLAMATION
*Managing Water in the West*

Technical Report ST-2019-1724-01

# SRH-Mesh User's Manual

**Sedimentation and River Hydraulics – Mesh, Version 1.0**



**U.S. Department of the Interior**
**Bureau of Reclamation**
**Technical Service Center**
**Sedimentation and River Hydraulics Group**                    September 2019

| REPORT DOCUMENTATION PAGE | | Form Approved<br>OMB No. 0704-0188 |
|---|---|---|

| **1. REPORT DATE** *(DD-MM-YYYY)*<br>9-29-2019 | **2. REPORT TYPE**<br>Numerical Model Documentation and User's Manual | **3. DATES COVERED** *(From - To)*<br>10-1-2016 to 9-29-2019 |
|---|---|---|

| **4. TITLE AND SUBTITLE**<br>SRH-Mesh 1.0 User's Manual<br> (Sedimentation and River Hydraulics – Mesh, Version 1.0) | **5a. CONTRACT NUMBER** |
|---|---|
| | **5b. GRANT NUMBER** |
| | **5c. PROGRAM ELEMENT NUMBER** |
| **6. AUTHOR(S)**<br>Jianchun "Victor" Huang | **5d. PROJECT NUMBER** |
| | **5e. TASK NUMBER** |
| | **5f. WORK UNIT NUMBER** |
| **7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)**<br>Sedimentation and River Hydraulics Group, Technical Service Center<br>Bureau of Reclamation<br>Denver Federal Center, Bldg. 67<br>PO Box 25007 (86-68240)<br>Denver, Colorado 80225-0007 | **8. PERFORMING ORGANIZATION REPORT NUMBER** |
| **9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)** | **10. SPONSOR/MONITOR'S ACRONYM(S)** |
| | **11. SPONSOR/MONITOR'S REPORT NUMBER(S)** |

**12. DISTRIBUTION/AVAILABILITY STATEMENT**

**13. SUPPLEMENTARY NOTES**

**14. ABSTRACT**
The Bureau of Reclamation's mesh generator SRH-Mesh 1.0 (Sedimentation and River Hydraulics – Mesh Version 1.0) is used to create a 2D mesh for hydraulic and sediment transport numerical. The model can input GIS images and shapefiles; delineate channel lines; and create triangle, quadrilateral, mixed, and Voronoi meshes. This manual provides step-by-step instructions for creating the 2D meshes.

**15. SUBJECT TERMS**
Mesh Generator, Alluvial Rivers, Triangle Mesh, Quadrilateral Mesh, Mixed Mesh, Voronoi Mesh

| **16. SECURITY CLASSIFICATION OF:** | | | **17. LIMITATION OF ABSTRACT**<br>none | **18. NUMBER OF PAGES**<br>77 | **19a. NAME OF RESPONSIBLE PERSON**<br>Jianchun Victor Huang |
|---|---|---|---|---|---|
| **a. REPORT**<br>U | **b. ABSTRACT**<br>U | **a. THIS PAGE**<br>U | | | **19b. TELEPHONE NUMBER** *(Include area code)*<br>303-445-2158 |

**Standard Form 298** (Rev. 8/98)
Prescribed by ANSI Std. Z39.18

**Technical Report ST-2019-1724-01**

# SRH-Mesh User's Manual

**Sedimentation and River Hydraulics–Mesh, Version 1.0**

**Prepared by**

_____

**Jianchun Victor Huang, P.E., Ph.D.**
**Bureau of Reclamation**
**Technical Service Center**
**Sedimentation and River Hydraulics Group**

**Peer Reviewed by**

_____

**Yong G. Lai, Ph.D.**
**Bureau of Reclamation**
**Technical Service Center**
**Sedimentation and River Hydraulics Group**

# Acronyms and Abbreviations

| | |
|---|---|
| 2D | two-dimensional |
| BCL | Boundary Condition Line Module |
| BC line | boundary condition line |
| Del | Delete |
| ft$^2$ | square feet |
| GEM | Geometry Module |
| GIS | Geographic Information System |
| IMG | Image Module |
| Ins | Insert |
| m$^2$ | square meters |
| MHG | Mesh Global Module |
| MHI | Mesh Individual Module |
| Reclamation | Bureau of Reclamation |
| Redist | Redistribution |
| SRH-Mesh | Sedimentation and River Hydraulics – Mesh |
| TIN | triangulated irregular network |
| Vert | vertex |

## Disclaimer

## Acknowledgements

# Contents

# 1. Introduction

## 1.1 SRH-Mesh Description

SRH-Mesh (Sedimentation and River Hydraulics – Mesh) is a graphical tool for generating unstructured two-dimensional (2D) meshes and is used for 2D hydraulics and sediment transport numerical simulations. It may import Geographic Information System (GIS) images and feature shapefiles; edit geometry; create triangle, quadrilateral, or mixed meshes; and create boundary condition lines. These generated meshes may be used in models such as the SRH-2D model which is the flow and mobile-bed sediment transport model developed at Reclamation. This manual describes the common functions in SRH-Mesh and provides step-by-step tutorials to create 2D meshes for river applications.

## 1.2 SRH-Mesh Capabilities

SRH-Mesh generates 2D meshes for numerical simulation in hydraulic engineering. Some of the model's capabilities are to:

- Import GIS geo-referenced images
- Import GIS geo-referenced shapefiles
- Draw, edit, and redistribute geometry lines as sides of meshes
- Create triangles, quadrilateral, mixed, or Voronoi meshes
- Combine meshes
- Assign bed properties
- Assign polylines for boundary conditions
- Assign boundary conditions
- Set model parameters

## 1.3 Acquiring SRH-Mesh

Requests may be sent directly to Reclamation's Sedimentation and River Hydraulics Group (Attention: SRH Support, Sedimentation and River Hydraulics Group, P.O. Box 25007 (86-68540), Denver, Colorado, 80225-0007).

SRH-Mesh is under continuous development and improvement.

## 1.4. Installation

Administrative rights are needed to install the program but are not required to run the program once installed. The program is designed to run on Window's platforms. However, the program has only been tested on Windows 10. Once installed, the program SRH-Mesh should be available through the start, programs menu under the SRH folder.

# 2. Graphical Interface

Presently, SRH-Mesh is a tool for generating triangle, quadrilateral, and mixed meshes along with the boundary condition lines (BC lines). It may be used by SRH-2D through the partial interface mode. The mesh generator, however, is under continuous development. In the future, SRH-Mesh will integrate with SRH-2D together so that the graphical interface of SRH-Mesh may be used to generate the mesh, assign boundary conditions, assign SRH-2D model inputs, run SRH-2D directly, and visualize model results.

SRH-Mesh interface window consists of six main regions as shown in Figure 2-1: Menu Bar, Table of Contents, Module Indicator, Tool Buttons, Status Line, and Workspace. This section describes the main functions of each region. Please use the tutorials in Section 5 to learn how these functions work.
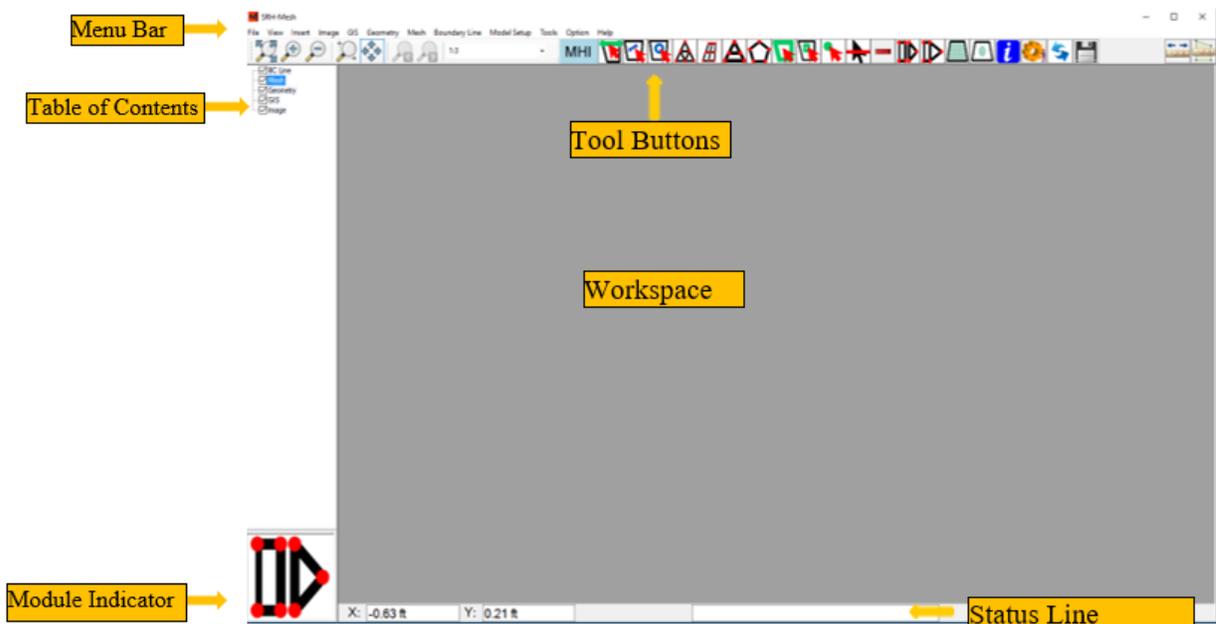


**Figure 2-1. SRH-Mesh's main window showing the six main regions: Menu Bar, Tool Buttons, Table of Contents, Workspace, Module Indicator, and Position Indicator.**

## 2.1. Menu Bar

SRH-Mesh's features are organized into the following menus (**Error! Reference source not found.**:
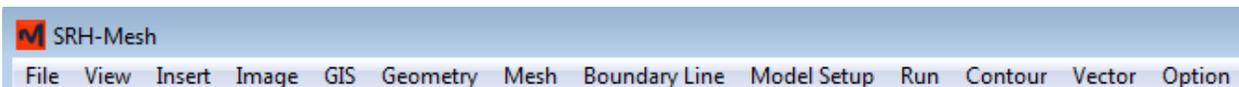


**Figure 2-2. SRH-Mesh's menu bar**

- **File:** control reading and writing of project files, and exiting SRH-Mesh
- **View:** control button sizes and shows coordinate projections

- **Insert:** insert text, shape, marker, north arrow, or scale bar

- **Image:** control image input

- **GIS:** control GIS shape files inputs and transfers GIS shape files into SRH-Mesh polylines

- **Geometry:** add, edit, or remove geometry

- **Mesh:** generate the mesh, assign the bed roughness, bed gradation zone, and initial boundary condition zone

- **Boundary Line:** add or delete boundary condition line

- **Model Setup:** create boundary condition lines and assign boundary conditions

- **Run:** run SRH-2D model (currently not available)

- **Contour:** add contour map from post-process file (currently not available)

- **Vector:** add vector map from post-process file (currently not available)

- **Option:** set display options

## *2.2. Module Indicator*

SRH-Mesh's module shows the set up and program modules.

**Setup module** is used to set up general model parameters, boundary conditions, initial conditions for unsteady simulations, and sediment size classes and bed gradations for mobile bed simulation.
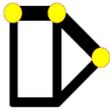
**BC line module** is used to create boundary and monitoring lines and assign boundary conditions.

**Mesh module** (individual mesh) is used to select lines to create mesh, assign bed roughness, etc.
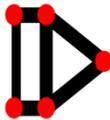
**Mesh module** (global mesh) is used to assign bed roughness, bed gradation zone, initial condition zone, etc.

Polylines delineate the sides of a mesh. The **Geometry module** is used to delineate polylines, edit nodes in polylines, redistribute nodes in polylines, and join polylines.

**GIS module** is used to input GIS shapefiles.

**Image module** is used to input images with GIS projections.

The status box currently provides the x and y coordinates of the mouse point.

Note that two other modules are under development:

**Vector module** will be used to create velocity vector map for 2D numerical simulation.
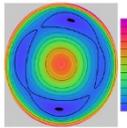
**Contour module** will be used to create contour map for 2D numerical simulation.

## 2.3. Table of Contents

SRH-Mesh's table of contents, as displayed in Figure 2-3, provides an easy click to turn the workspace contents on or off. The first layer of table of contents shows the modules available in the SRH-Mesh program. When you click a module in the table of contents, the program will switch to the corresponding module.

If there is a plus + sign to the left of the contents, then user-created subfiles are available under that content line. For example, if you add four GIS shapefiles to the program, then there will be four sub-contents with the shapefile names under GIS, and you can switch each shapefile on and off. You can expand the table of contents on the plus sign to the left of the contents to show the sub-contents.
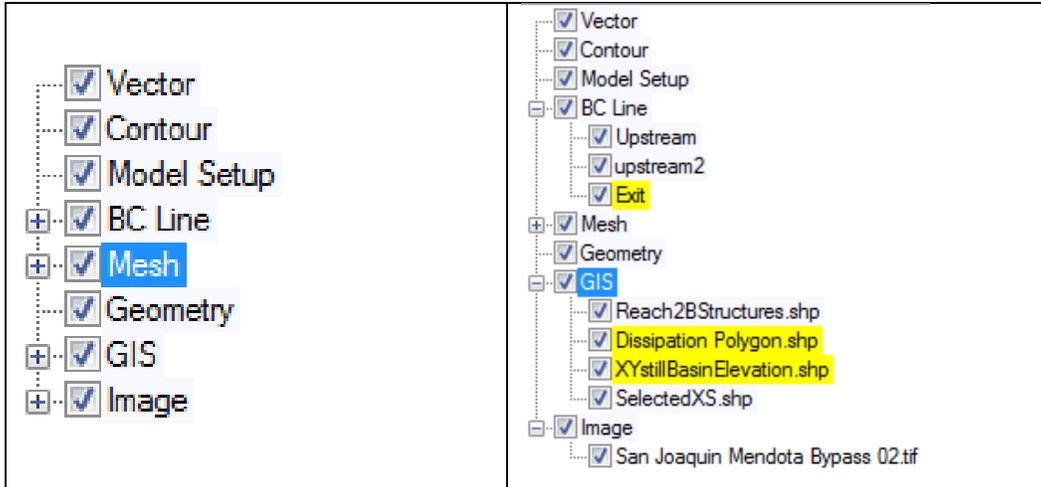
**Figure 2-3. SRH-Mesh's table of contents.**

## 2.4. Tool Buttons

Tool buttons are just under the menu. When a module is clicked in the table of contents, the tool buttons are updated for that module. Eight zoom buttons are followed by a serial of buttons for each module.

### 2.4.1. Zoom Buttons

The same zoom tools for all four modules followed by different buttons for each module (Figure 2-4.



**Figure 2-4. Zoom buttons.**

Each button provides a different function:

| | |
|---|---|
|  | Zoom to the map's extent |
|  | Zoom into map |
|  | Zoom out of map |
|  | Specify viewport by mouse selection |
|  | Drag the map's content around and scroll by mouse wheel |
|  | Zoom to previous viewport |

|  | Zoom to next viewport |
|---|---|
| 1:1000 | Set the view scale |

## 2.4.2. Image Module Buttons

When you click the Image Module in the table of contents, the Image Module toolbar will appear next to the zoom buttons (Figure 2-5):

IMG

**Figure 2-5. Command buttons for Image Module.**

Each button provides a different function.

| IMG | Label to show image module |
|---|---|
| | Add an image from file |
| | Remove selected image from project |
| *i* | Display general information for selected image geometry in the context table (file name and source, projection, size, and envelope) |

Currently SRH-Mesh can input images in tif, img, and ecw formats. All input images should be GIS referenced.

## 2.4.3. GIS Module Buttons

When you click the GIS Module in the table of contents, the GIS Module toolbar will appear next to the zoom buttons (Figure 2-6):

GIS

**Figure 2-6. Command buttons for GIS Module.**

Each button provides a different function:

| GIS | Label to show GIS module |
|---|---|
| | Add a GIS shapefile to the project |
| | Remove selected GIS shapefile from project |

Transfer selected GIS geometry in the context table into SRH-Mesh geometry

Display general information of selected GIS geometry in the context table (file name, projection, type, size, and envelope)

Format display of the geometry (line width and color, point size and color, polygon fill color)

## 2.4.4. Geometry Module Buttons

### 2.4.4.1 Create and Edit Geometry

When you click the Geometry Module (GEM) in the table of contents, the Geometry Module Toolbar will appear next to the zoom buttons:

**Figure 2-7. Command buttons for the Geometry Module.**

Each button provides a different function.

GEM      Label to show geometry module.

Select a line.

Remove all selections.

Add geometry by mouse click.

Edit geometry lines.

Move/rotate geometry lines

Merge/split geometry lines.

Split geometry lines at all intersections.

Show geometry information. When no line is selected, a window shows information about all lines: total line numbers, maximum line length, minimum line length, maximum number of vertexes, and minimum number of vertexes. When a line is selected, a window shows information about the selected line and coordinates of the line.

7

Format display of the geometry (line and node size and color).

### 2.4.4.2 Edit Polylines

When you click the edit geometry button ✎, the Edit Tools Module toolbar
(Figure 2-8), which is used to edit the selected polylines (Sections 3.3.4 to
3.3.11).



**Figure 2-8. Command buttons for the Edit Tools Module to edit lines.**

Each button provides a different function.

| | |
|---|---|
| ▶ | Select the geometry line(s) to be edited. Press Ctrl to add or remove the selection to previous selections. Press Alt to add the selection to previous selections—even if that selection is already in the previous selections. |
| ✖ | Deselect all features. |
| Delete | Delete selected polyline(s). |
| Redis | Redistribute selected polyline(s) and input the number of vertexes. |
| Edit Vert | Edit the selected vertex by input x and y coordinates. Coordinates can by input directly or by an equation. For example, if you input "=x-100" in the x textbox, the selected vertex will be moved by 100 units (e.g., feet) to the left. |
| Move Vert | Move a vertex: use the mouse to drag the selected vertex into a new location. |
| Ins Vertex | Insert a new vertex by clicking on it. |
| Del Vert | Delete a vertex by clicking on it. |
| 💾 | Save edited geometry. |
| 💾 | Save edited geometry and exit the Edit Geometry Module. |

🚫         Exit the Edit Geometry Module without saving the edits.

### *2.4.4.3 Move or Rotate Polylines*

When you click the Move/Rotate button 👆, the program enters the Move/Rotate Module toolbar (Figure 2-9, which are used to move, rotate and scale the selected polylines (Sections 3.3.12 to 3.3.15).



**Figure 2-9. Command buttons for moving, rotating, and scaling geometry.**

Each button provides a different function.

| | |
|---|---|
| ▲ (cursor) | Select the geometry to be moved/rotated. Press Ctrl to add or remove the selection to previous selections. Press Alt to add the selection to previous selections— even if that selection is already in the previous selections. |
| ▲ (cursor with line) | Deselect all features. |
| Move | Move selected polylines by inputting x and y distances, or use the + or – key on your keyboard to shift the selected polylines to the left (-) or right (+). You can keep the original polylines. |
| Move with Mouse | Move the selected polyline by dragging it to a new location. |
| Select Rotate Point | Select the rotate point. |
| Rotate/Scale | Rotate/scale selected polylines with respect to the rotate point. The rotate is input in a range from $0^o$ to $360^o$. You can keep the original polylines. |
| 💾 | Save edited geometry. |
| 💾➡ | Save edited geometry and exit the Move/Rotate Module. |
| 🚫 | Exit the Move/Rotate Module without saving the edits. |
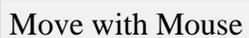
### 2.4.4.4 Merge or Split Polylines

When you click the Merge/Split button [image], the program enters the Merge Tools Module (Figure 2-10,) which is used to merge, split, group, and ungroup the selected polylines (Sections 3.3.16 to 3.3.20).



**Figure 2-10. Command buttons for merging, splitting, and grouping geometry lines.**

Each button provides a different function.

| | |
|---|---|
| [arrow icon] | Select the geometry lines to be merged/grouped. Press Ctrl to add/remove selections, press Alt to add selections. |
| [arrow with red line icon] | Deselect all features. |
| Merge | Merge selected geometry into one feature. |
| Split at Vertex | Split the selected geometry by clicking a vertex. |
| Split at Intersection | Split the selected geometry at all intersection(s). |
| Group | Group selected geometries into one group. The polylines in the same group can be treated as one of four sides in a quadrilateral mesh. |
| Ungroup | Ungroup selected geometries into one group. The polylines in the same group can be treated as one of four sides in a quadrilateral mesh. |
| [save icon] | Save geometry. |
| [save and exit icon] | Save edited geometry and exit the Merge Tools Module. |
| [prohibited icon] | Exit the Merge Tools Module without saving the edits. |

## 2.4.5. Individual and Global Mesh Module Buttons

SRH-Mesh can create four types of meshes:

- **Triangle** meshes contain only triangles from Delaunay triangulations resulting from arbitrary boundaries and inside lines or holes.

- **Quadrilateral** meshes are created as structure grid using four boundary lines (which can be combined from more than four lines). If there are uneven vertexes between opposite side lines, then a triangle cell can be inserted to transit from one side to another side of domain.

- **Mixed** meshes are created by merging neighbor triangle cells into quadrilateral cells. The resulting mesh contains most quadrilateral cells with limited triangle cells that can't be merged with other cells, or if it is merged the quadrilateral cell is too irregular.

- **Voronoi** meshes can be created, but if one Voroni mesh is created, then the final mesh should contain all Voronoi cells.

When you click the Mesh Module (M), the Mesh Module toolbar will appear next to the zoom buttons (Figure 2-11):



**Figure 2-11. Command buttons for the Mesh Module.**

Each button provides a different function. In the model, all buttons are presented in a single line. Some of the buttons are available only when needed.

| | |
|---|---|
| MHG | Label to show global mesh module. |
| MHI | Label to show individual mesh module. |
| | Select a single side or multiple sides for mesh generation. |
| | Select an inside line or multiple insider lines for mesh generation. |
| | Select a single or multiple hole line(s) for mesh generation. |
| | Create a triangle mesh with selected sides, inside lines (optional), and hole lines (optional). |

Create a quadrilateral mesh with selected side lines.

Create a mixed mesh with selected sides, inside lines (optional), and hole lines (optional).

Create a Voronoi mesh with selected sides, inside lines (optional), and hole lines (optional).

Select a single or multiple mesh domain(s).

Select a single or multiple mesh cell(s).

Select a single or multiple mesh vertex(es).

Remove all sections.

Remove selected mesh(s).

Show individual meshes.

Show global mesh.

Change the bed roughness zone in selected mesh(s).

Change select bed roughness zone in selected mesh cell(s).

Change the sediment gradation zone in selected mesh(es). This button is only available for the morphology solver.

Change the selected sediment gradation zone in selected mesh cells. This button is only available for the morphology solver.

Change initial zone in selected mesh(s). This button is only available for the unsteady flow or morphology solver with an initial condition input at each zone.

Change initial zone in selected mesh cells. This button is only available for the unsteady flow or morphology solver with an initial condition input at each zone.

Display the mesh information regarding cell number, vertex number, vertex number on the boundary, area, maximum area, minimum area, maximum angle, and minimum angle of cells.

Set mesh display properties. You can choose the size and color of mesh lines, boundary lines, mesh fills for different zones of bed roughness, bed gradation, and initial condition.

Refresh the mesh after the elevation was changed. Mesh elevation is changed on mesh vertexes in each individual mesh, and the global mesh elevation will be updated only when the refresh button is clicked.

Save the created mesh in .srhm format.

## 2.4.6. Boundary Condition Line Module Buttons

When you click the BC Line Module in the table of contents, the BC Line Module Toolbar will appear next to the zoom buttons (Figure 2-12).
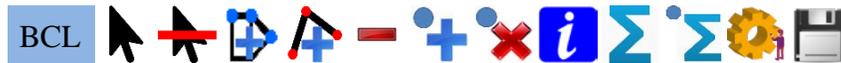


**Figure 2-12. Command buttons for the BC Line Module.**

Each button provides a different function.

BCL — Label to show BC Line Module.

Select a boundary condition line to be removed or to assign boundary conditions.

Deselect all features including boundary condition lines.

Add a boundary condition line.

Add an internal monitoring line.

Remove boundary condition lines or monitoring lines.

Add a monitoring point.

Remove a monitoring point.

Display a summary of all boundary condition lines.

Display a summary of boundary condition lines (name, boundary condition type, boundary condition index). The unused boundary conditions are also shown.

Display a summary of monitoring point(s).

Set the boundary condition line display properties. You can choose the size, color, and pattern for the boundary condition lines, and the boundary condition line label font.

Save the created mesh in .srhm format.

All points of boundary condition lines are located on the external boundary of the final global mesh, and all points of monitoring lines are located inside of the global mesh. Monitoring lines can also be used to setup internal boundary conditions. To create the boundary condition lines, first click the add a boundary condition line button and then use the mouse to select vertexes. SRH-Mesh will add boundary vertexes between the selected vertexes to create the boundary condition line. To create the insider monitoring lines, first click the add an internal monitoring line button and then use the mouse to select inside vertexes. To finish adding vertexes to a boundary condition line or a monitoring line, double-click the last vertex as the end of the line. SRH-Mesh will add vertexes between the selected vertexes to create a monitoring line that has the shortest distance between the selected vertexes. To delete a created boundary condition line or monitoring line, first click remove boundary condition lines or monitoring lines button , then click the line you want to delete. To add a monitoring point, first click the add a monitoring point button , then click the vertex as a monitoring point. To remove a monitoring point, first click remove a monitoring point button , then click the monitoring point.

To assign or edit the boundary condition for a boundary condition line, first select a boundary condition line. Then right-click to bring up the context menu strip

where you can edit the current boundary condition, add a new boundary condition, assign an existing boundary condition to the line, or delete the boundary line. To select a boundary condition line, you can use the select button ➤ and then click the line directly, or you can double-click the line in the table of contents.

## 2.5. Status Line

The status line includes textboxes for x, y, and z coordinates as well as a message box for program status (Figure 2-13). While the mouse is moving in the workspace, the mouse coordinates are listed in the x and y textboxes. When some mesh vertexes are selected, the first mesh vertex coordinates are listed in the x, y, and z textboxes. The user can also change the elevation of selected mesh vertexes directly by inputting new z coordinate in the text box. The message box provides the program's running information.

X: 6600037.49 ft    Y: 1745307.63 ft    Z: 100    Elevation is updated.

**Figure 2-13. Status line.**

# 3. Modules

## *3.1. Image Module*

Click Image in the table of contents to switch to the Image Module toolbar (Figure 3-1).

The icon for the Image Module will appear in the lower left corner of the workspace .

IMG

**Figure 3-1. Command buttons for the Image Module.**

### 3.1.1. To Add an Image

1. Click the add button
2. Select an image file from a file browser
3. Click the open button

If you are using an image exported from GIS, perform the following steps to ensure that the image is readable and can be correctly projected in SRH-Mesh:

> In ArcMap
> Select File>Export Map
> Save as type: TIFF(*tif)
> Make sure "Write World File" is selected.
> Save
>
> In Arc Cetology
> Right-click the created image file (*tif)>Properties
> Selects Spatial Reference>Edit
> Select or add the corrected coordinate system.

### 3.1.2. To Remove an Image

1. Select the image from the table of contents under Image
2. Click the remove button

### 3.1.3. To Display an Image Information

1. Select the image from the table of contents under Image
2. Click the information button

## *3.2. GIS Module*

Click GIS in the table of contents to switch to GIS Module toolbar (Figure 3-2). The icon

for the GIS Module will appear in the lower left corner of the workspace.

GIS

**Figure 3-2. Command buttons for the GIS Module.**

### 3.2.1. Add an GIS Shapefile

1. Click the add GIS file button

2. Select the GIS file from a file browser

3. Click the open button

### 3.2.2. Remove a GIS

1. Select the GIS Module toolbar from the table of contents under GIS

2. Click the remove GIS file button

### 3.2.3. Display a GIS Shapefile Information

1. Select the GIS Module toolbar from the table of contents under GIS

2. Click information button

### 3.2.4. Convert a GIS Shapefile into SRH-Mesh Geometry

1. Select a GIS shapefile with a yellow background color from the table of contents under GIS. If the background of the shapefile name in the table of contents is white, then it has already been converted into SRH-Mesh Geometry. If you want to convert the shapefile again, you need to remove the GIS shapefile and add it into the project again.

2. Click the transfer GIS geometry button

### 3.2.5. Display a GIS Shapefile Information

1. Select a GIS shapefile from the table of contents under GIS

2. Click the information button

### 3.2.6. Reset GIS Display Options

1. Click the format display button

2. Set all display options:
   a. Line style, width, and color
   b. Point style, size, and color
   c. Polygon fill color

3. Click the yes button to reset the GIS display options or the no button to cancel the action

## *3.3. Geometry Module*

Click Geometry in the table of contents to switch to Geometry Module toolbar (Figure 3-3). The icon for the Geometry Module will appear in the lower left corner of the workspace ⌐.

GEM ▶ ▶ ╋ ✎ ♨ ⚙ / 𝒊 ⚙

**Figure 3-3. Toolbar for editing geometry.**

Note that lines can be created from the points that are part of the SRH-Mesh geometry. However, they can't be edited in the SRH-Mesh.

### 3.3.1. Select a Geometry Line

1. Click the select line button ▶

2. Mouse to the line(s) you want

   a. Click to select a line

   b. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s)

   c. Click and drag to select all lines in the drag box

### 3.3.2. Deselect All Lines

Click the deselect all button ▶ to deselect all lines in the Geometry, Mesh, and Boundary Condition Line Modules.

### 3.3.3. Add a Geometry Line Using the Mouse

1. Select Geometry>Snap on (off) to set the option if you want to snap the line to a vertex of another line or a geometry point (optional).

2. Select Geometry>Snap Distance = ## Pixels to set the snap distance (optional). A form will appear to set the snap distance. If the distance is greater than the snap distance on the screen, then the new vertex would not be snapped to any vertex or point.

3. Click the add line button ╋.

4. Click to add a line in the workspace.

5. You can use the zoom buttons: zoom to the map's extent , zoom into the map , zoom out of the map , zoom to a previous viewport , and zoom to the next viewport . You may also click and drag to move the map around and use mouse wheel to zoom into and out of map.

6. Double-click to finish adding a line.

## 3.3.4. Add Points from a Text File

Select Geometry> Input Text File as Points. A form will appear (Figure 3-4).
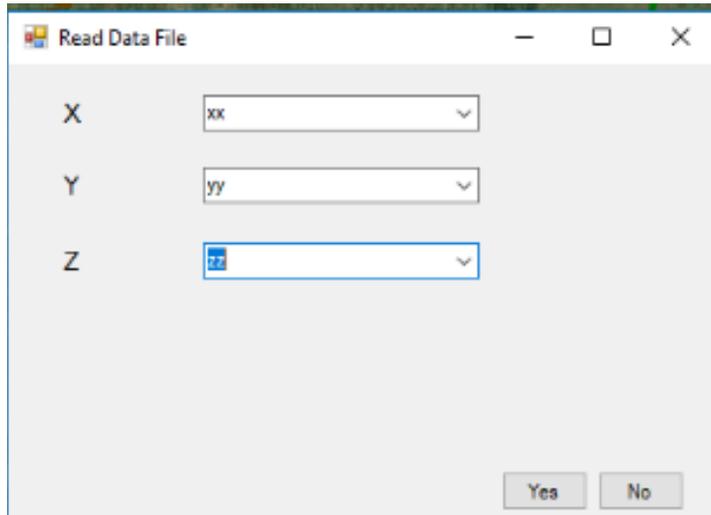


**Figure 3-4. Input points from text file.**

1. Select a file with extension .csv, .txt, or .dat from a file browser

2. Input the parameters

    a. X: column for the x coordinate

    b. Y: column for the y coordinate

    c. Z: column for the z coordinate (optional)

3. Click the yes button to input the text file or the no button to cancel the action

## 3.3.5. Add Lines from a Text File

1. Select Geometry>Input Text File as Polylines. A form will appear (Figure 3-5).
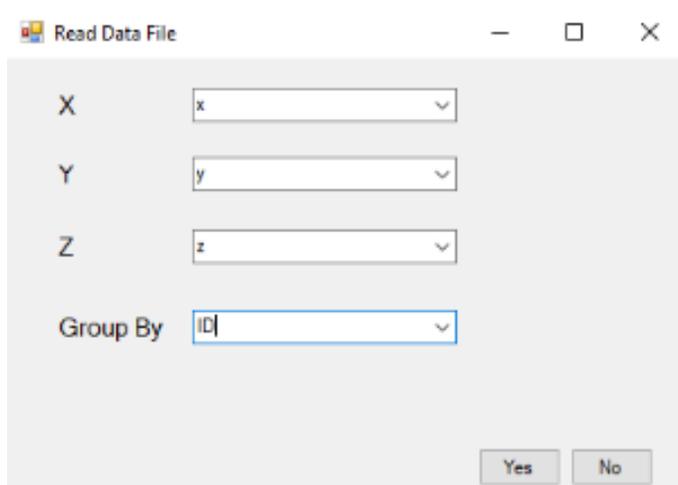


**Figure 3-5. Input lines from text file.**

2. Select a file with extension .csv, .txt, or .dat from a file browser

3. Input the parameters

    a. X: column for the x coordinate

    b. Y: column for the y coordinate

    c. Z: column for the z coordinate (optional)

    d. Group By: Select from the drop down menu. For example, selecting to group by ID would group all the points with same ID would be in a single line

4. Click the yes button to input the text file or the no button to cancel the action

## 3.3.6. Delete a Line

1. Click the edit line button ✎ to bring up the editor toolbar if the editor controls (Figure 3-6) are not shown.



**Figure 3-6. Toolbar for editing lines.**

2. Click the select a line button ➤ in the editor toolbar and select the line(s) to be deleted.

3. Click the delete button Delete to delete selections.

4. Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button 💾 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

## 3.3.7. Redistribute Line

1. Click the edit line button ✎ to enter editor toolbar if editor controls (Figure 3-6) are not shown.

2. Click the select line button ➤ in the editor toolbar and select the line(s) to be redistributed.

3. Click the redistribute button Redist . A form will appear (Figure 3-7).
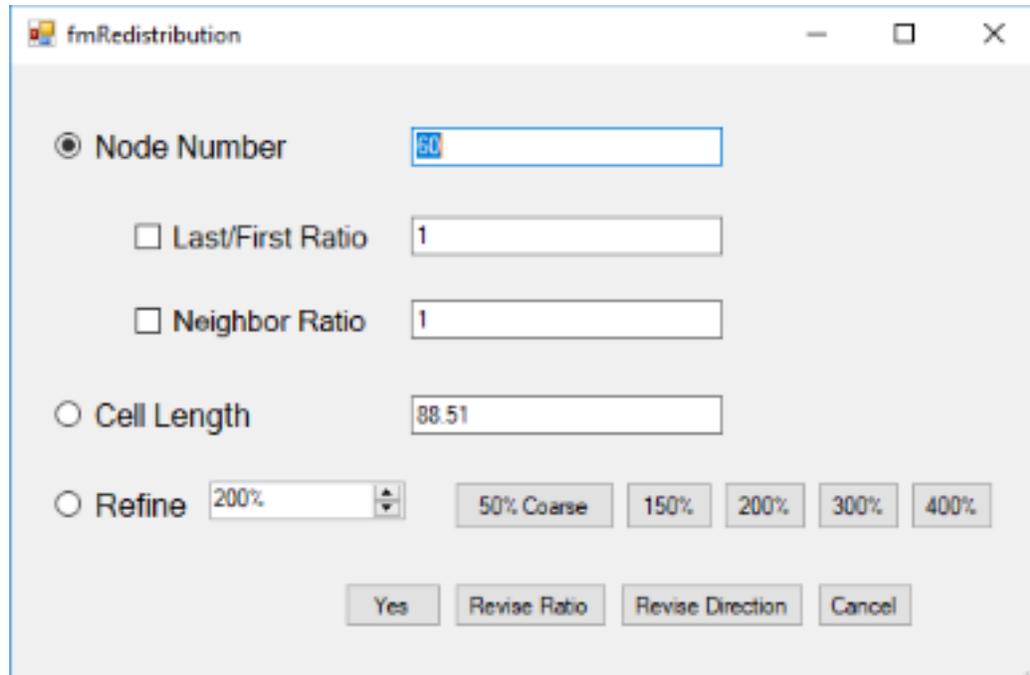
**Figure 3-7. Geometry line redistribution form.**

4. Input the parameters.

   a. **Node Number**: total number of vertexes.

   b. **Last/First Ratio:** the ratio of the last segment length to first segment length.

   c. **Neighbor Ratio:** the ratio of length of segment $i+1$ to length of segment $i$.

   d. **Cell Length:** cell length of each segment. The line will be equally distributed.

   e. **Refine:** The line will be refined according to input ratio.

5. Options to redistribute the line(s).

   a. Click the yes button to perform the defined redistribution task or the no button to cancel the action

   b. Click the revise ratio button to recalculate the Last/First or Neighbor Ratio as the reciprocal of the ratio and perform the redistribution task

   c. Click the revise direction button to redistribute the line according to input parameters, then reverse the direction of the line

6. Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button to save edited geometry and exit the editor  toolbar or click the cancel button 🚫 to exit editor  toolbar without saving.

## 3.3.8. Edit a Vertex in a Geometry Line

1. Click the edit line button ✏ to enter editor  toolbar if editor controls (Figure 3-6) are not shown.

2. Click the select line button ⬆ in the editor toolbar and select the line to be edited.

3. Click the edit vertex button | Edit Vertex | .

4. Click to select the vertex from the line to be edited.

5. In the x, y, or z text box in the Status Line, input the new x, y, or z value.

   a. Input x, y, or z coordinate in the format of ####.## (e.g., 1234567.89)

   b. Input relative coordinates in the x, y, or z textbox (e.g., if you input "=y-100" as y coordinate, then the y coordinate will be decreased by 100)

7. Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button 💾 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

## 3.3.9. Move a Vertex

1. Click the edit line button ✎ to enter editor toolbar if editor controls (Figure 3-6) are not shown.

2. Click the select line button ⬆ in the editor toolbar and select the line to be edited.

3. Click the move vertex button | Move Vert | .

4. Click to select the vertex to be moved and drag it to the target location.

5. Continue to edit geometry lines, or the save button 💾 to save edited geometry, or click the save and exit button 💾 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

## 3.3.10. Insert a Vertex

1. Click the edit line button ✎ to enter editor toolbar if editor controls (Figure 3-6) are not shown.

2. Click the select line button ⬆ in the editor toolbar and select the line to be edited.

3. Click the insert vertex button | Ins Vert | .

4. Click on the point where you want to insert the new vertex. The new vertex can only be inserted into the original line.

5. Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button 💾 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

### 3.3.11. Delete a Vertex

1.  Click the edit line button ✎ to enter editor  toolbar if editor controls (Figure 3-6) are not shown.

2.  Click the select line button ➤ in the editor  toolbar and select the line to be edited.

3.  Click the delete vertex button ⟨Del Vert⟩.

4.  Click on the vertex to be deleted.

5.  Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button 💾 to save edited geometry and exit the editor  toolbar or click the cancel button 🚫 to exit editor  toolbar without saving.

### 3.3.12. Move a Line

1.  Click the move line button 🖐 in the Geometry Module toolbar (Figure 3-8) to enter the move/rotate toolbar if this toolbar is not shown (Figure 3-8).



**Figure 3-8. Toolbar for moving, rotating, and scaling geometry.**

2.  Click the select line button ➤ in the editor toolbar and select the line(s) to be moved.

3.  Click the move button ⟨Move⟩. A form will appear (Figure 3-9).

**Figure 3-9. Form to move a line.**

4.  Input the parameters.

    a.  **X Y Directions:** Distances along the x and y coordinates to move the line.

    b.  **Node Coordinates:** These coordinates define how much the line will be shifted parallel to the original line. A positive value would shift the line to the right direction and a negative value would shift the line to the left direction.

    c.  **Keep the Original:** This determines if you want to move the line to a new direction while also keeping the original line.

5.  Click the yes button to perform the action or the no button to exit the form.

6.  Continue to edit geometry lines, or click the save button ⎘to save edited geometry, or click the save and exit button ⎘ to save edited geometry and exit the editor  toolbar or click the cancel button ⊘ to exit editor  toolbar without saving.

## 3.3.13. Move a Line with the Mouse

1.  Click the move line button 🖐 to enter move/rotate  toolbar if the move/rotate controls (Figure 3-8) are not shown.

2.  Click the select line button ➘ in the editor  toolbar and select the line(s) to be moved.

> Move with Mouse

3.  Click the move with mouse button and drag the line to a new location.

4. Continue to edit geometry lines, or click the save button ![save] to save edited geometry, or click the save and exit button ![save and exit] to save edited geometry and exit the editor toolbar or click the cancel button ![cancel] to exit editor toolbar without saving.

## 3.3.14. Select a Rotate Point to Rotate/Scale a Line

When you want to rotate a line or scale a line, first select a rotate point. You can move or scale multiple lines at the same time with the same rotate point. If you do not select a rotate point, the center of all lines will be used as a rotate point. The x and y coordinates can be modified in the Rotate/Scale form by inputting new values.

1. Click the move line button ![move] to enter the move/rotate toolbar if move/rotate controls (Figure 3-8) are not shown.

2. Click the select-line button ![select] in the editor toolbar and select the line(s) to be moved.

3. Click the rotate point button Rotate Point .

4. Click a vertex in the line as the rotate point.

5. Rotate/Scale a Line (Section 3.3.15).

### 3.3.15. Rotate/Scale a Line

When you want to rotate a line or scale a line, select a rotate point (3.3.14).

1. Click the move ✍ button in the Geometry Module toolbar (Figure 3-3) to enter the move/rotate toolbar if the move/rotate controls are not shown (Figure 3-8).

2. Click the rotate/scale button ⎡Rotate/Scale⎤. A form will appear (Figure 3-10).
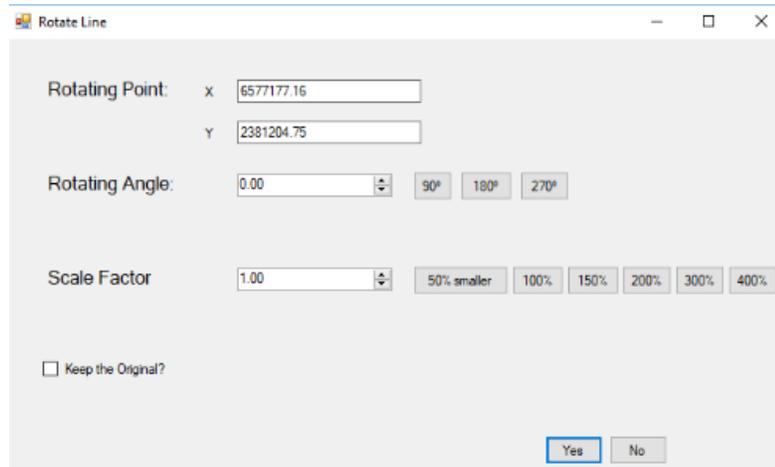


**Figure 3-10. Rotate or scale a line.**

3. Input the parameters.

   a. **Rotating point x and y:** x and y coordinate where the line(s) would be rotated or scaled.

   b. **Rotating angle:** angle with which the line(s) would be rotated. A negative value between -180º to 0º (clockwise) or a positive value from 0 to 360º (counterclockwise) is allowed. Quick buttons of 90º, 180º, and 270º are provided.

   c. **Scale factor:** value to scale the line(s). A value between 0.10 to 10 is allowed. Quick buttons of 50%, 100%, 150%, 200%, 300%, and 400% are provided.

   d. **Keep the original:** This determines if you want to rotate/scale the line(s) while also keeping the original line.

4. Click the yes button to perform the action or the no button to cancel the action.

5. Continue to edit geometry lines, or click the save button 💾 to save edited geometry, or click the save and exit button 📤 to save edited geometry and exit the editor  toolbar or click the cancel button 🚫 to exit editor  toolbar without saving.

### 3.3.16. Merge Lines

1. Click the merge/group button 🔗 to enter merge/group  toolbar if merge/group controls (Figure 3-11) are not shown.

**Figure 3-11. Toolbar for merging, splitting, and grouping geometry.**

2. Click the select line button ↖ in the editor toolbar and select the line(s) to be merged.
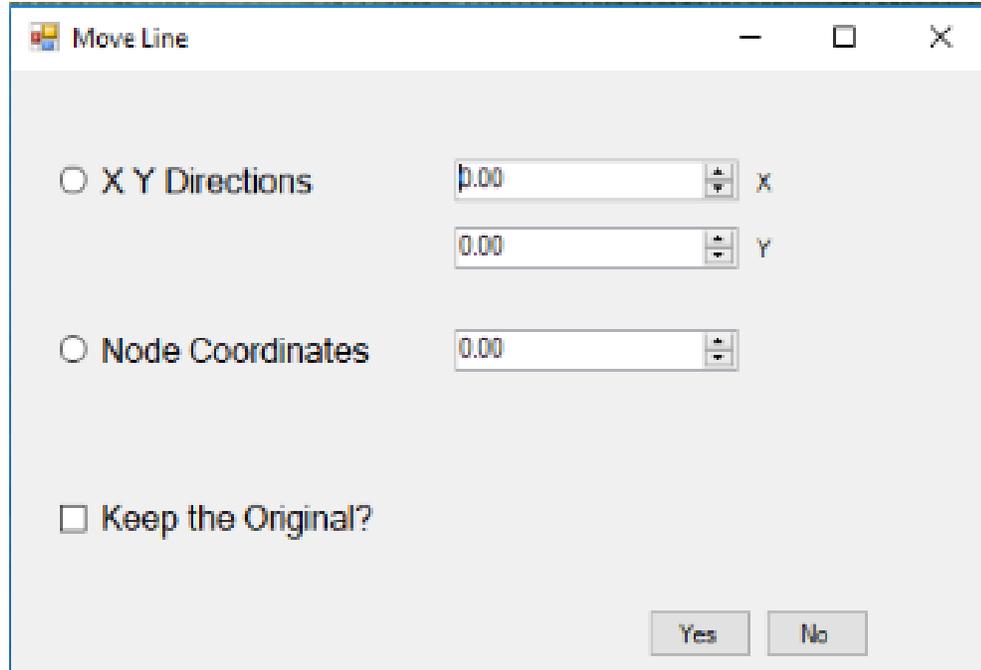
3. Click the merge button Merge .

4. Continue to edit geometry lines, or click the save button 🖫 to save edited geometry, or click the save and exit button 🖫 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

### 3.3.17. Split a Line

1. Click the merge/group button 🔧 to enter merge/group toolbar if merge/group controls (Figure 3-11) are not shown.

2. Click the select line button ↖ in the editor toolbar and select the line(s) to be merged.

3. Click the split button Split .

4. Click a vertex where you want to split the line.

5. Continue to edit geometry lines, or click the save button 🖫 to save edited geometry, or click the save and exit button 🖫 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

### 3.3.18. Split lines at Intersection(s).

1. Click the merge/group button 🔧 to enter merge/group toolbar if merge/group controls (Figure 3-11) are not shown.

2. Click the select line button ↖ in the editor toolbar and select the line(s) to be merged.

3. Click the split at intersection button Split at Intersection .

4. Continue to edit geometry lines, or click the save button 🖫 to save edited geometry, or click the save and exit button 🖫 to save edited geometry and exit the editor toolbar or click the cancel button 🚫 to exit editor toolbar without saving.

### 3.3.19. Group Lines for a Quadrilateral Mesh

Only four lines are required for four sides of a quadrilateral mesh. If there are more than four lines selected as a boundary for a mesh, SRH-Mesh automatically detects the shape of the boundary and determines how to group several lines into one side of the mesh. If you want to use another way to composite a side of a mesh, you can group these connected lines by setting a same group number for the connected lines. Following these steps to group lines.

1. Click the merge/group button ![icon] to enter merge/group toolbar if merge/group controls (Figure 3-11) are not shown.

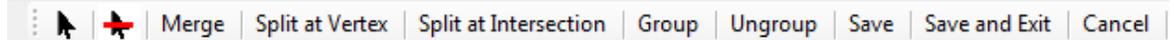2. Click the select-line button ![icon] in the editor toolbar and select the line(s) to be merged.

3. Click the group button Group .

4. Continue to edit geometry lines, or click the save button ![icon] to save edited geometry, or click the save and exit button ![icon] to save edited geometry and exit the editor toolbar or click the cancel button ![icon] to exit editor toolbar without saving.

### 3.3.20. Ungroup Lines for a Quadrilateral Mesh

If you do not like the way that you grouped the lines, following these steps to ungroup lines.

1. Click the merge/group button ![icon] to enter merge/group toolbar if merge/group controls (Figure 3-11) are not shown.

2. Click the select-line button ![icon] in the editor toolbar and select the line(s) to be group.

3. Click button Ungroup .

4. Continue to edit geometry lines, or click the save button ![icon] to save edited geometry, or click the save and exit button ![icon] to save edited geometry and exit the editor toolbar or click the cancel button ![icon] to exit editor toolbar without saving.

### 3.3.21. Split Lines at all Line Intersections

Click the split-lines button ![icon] .

### 3.3.22. Display Geometry Information

1. Select the GIS shapefile from the table of contents under the GIS menu item

2. Click the information button 𝒊 to display geometry information

   a. If no line is selected, information about all lines is displayed (**Error! Reference source not found.**)

       • **Line Number:** total number of lines

       • **Max Length:** the length of line with maximum length

       • **Min Length:** the length of line with minimum length

       • **Max Vertex Number:** number of vertexes in the line with maximum vertexes

       • **Min Vertex Number:** number of vertexes in the line with minimum vertexes

   b. If you select a line, a form will appear (Figure 3-13).

   c. Click the Info and Coordinate tabs to switch between line information and coordinates. The Coordinate tab provides x, y, and z coordinates. The Info tab contains:
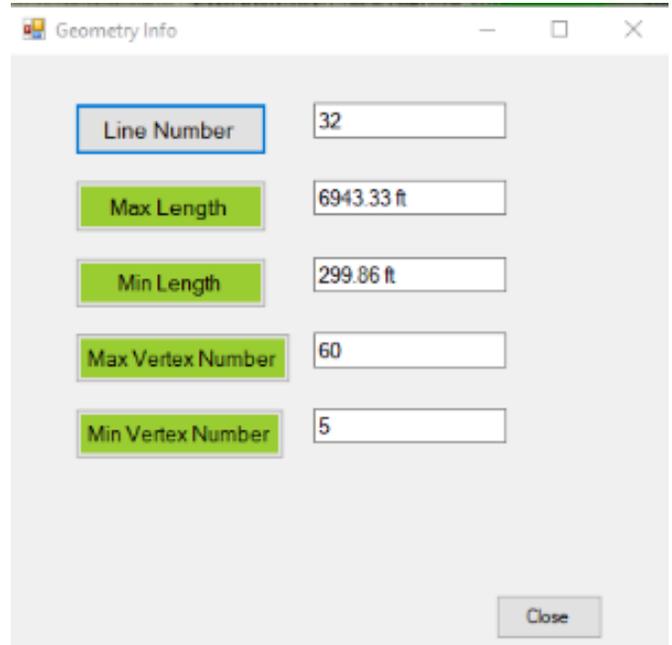


**Figure 3-12. Information for all lines.**
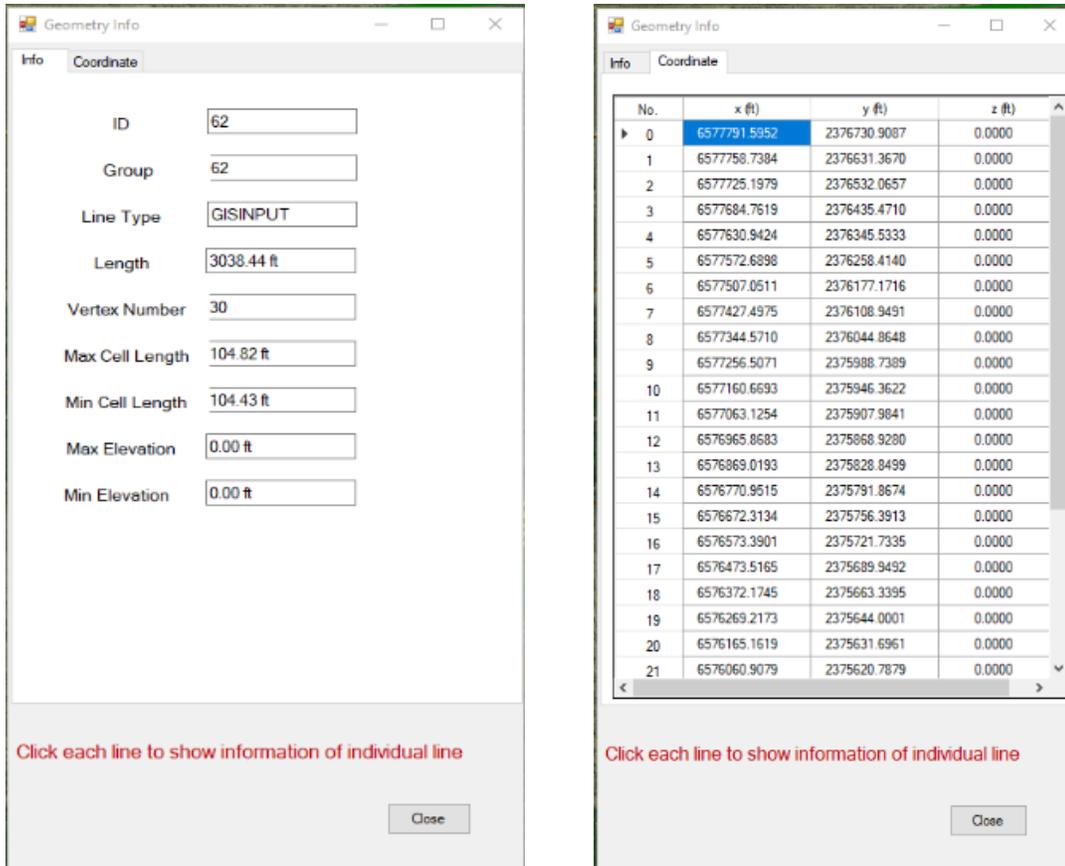
Line Information                                          Line Coordinates

**Figure 3-13. Information for a single line.**

- **ID:** line ID

- **Group:** line group number

- **Line Type:** GIS input lines, created lines, large wood structure line, large wood structure boundary line, or large wood structure linkage line

- **Length:** length of line

- **Max Cell Length:** maximum cell length

- **Min Cell Length:** minimum cell length

- **Max Elevation:** maximum elevation

- **Min Elevation:** minimum elevation

3. Click the close button to close the form.

### 3.3.23. Display Geometry Summary

1.  Select a GIS shapefile from the table of contents under GIS

2.  Click the summary button $\Sigma$ to display all of the line information (Figure 3-13)

3.  Click the close button to close the form

### 3.3.24. Set Geometry Display Options

1.  Click the set-display button . A form will appear (Figure 3-14).
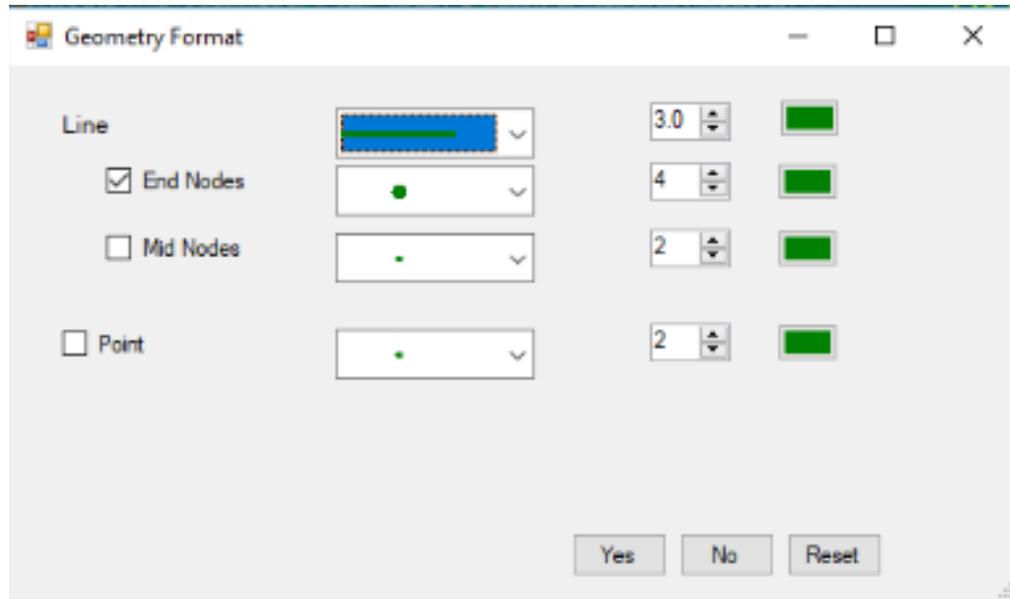


**Figure 3-14. Geometry format.**

2.  Input parameters:

    a.  Line style, width, and color

    b.  End node style, size, and color

    c.  Middle node style, size, and color

    d.  Point geometry style, size, and color

3.  If needed, click the reset button to return default setting.

4.  Click the yes button to save the geometry display options or the no button to cancel the action.

### 3.3.25. Create Mesh from all Geometry

Select Geometry>Create Mesh with All Geometry Lines or Mesh>Create Mesh with All Geometry Lines.

### 3.3.26. Export All Geometry Lines into a GIS Shapefile

Select Geometry>Export to GIS ShapeFile and the shapefile is saved as
C:\TEMP\ExportShapefile.shp.

### 3.3.27. Insert a Simplified Large Wood Structure

SRH-Mesh enables you to add six types of pre-defined simplified large wood structures
(Figure 3-15).
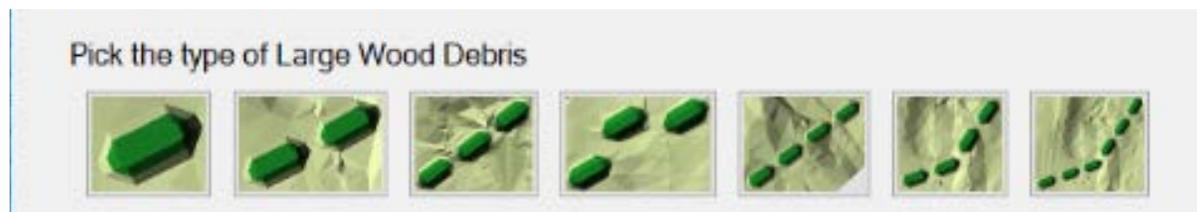


**Figure 3-15. Six types of pre-defined simplified large wood structure.**

Follow the following steps to add a large wood structure.

1. Select Geometry>Insert Simplified Large Wood Structures.

2. Click a point in the workspace where you want to insert the beginning of the
   structure, hold and drag the mouse to a new point where you want to insert the end
   of the structure. The direction from the first to second point represents the
   orientation of the large wood structure. The middle point of your mouse "hold and
   drag" represents the center of the large wood structure. A form will appear (
   Figure 3-16).

**Figure 3-16. Form to pick the type of simplified large wood structure and set the dimension of the structure.**

3. Select one of the six types of pre-defined simplified large wood structures in the form. The form will change its representing figure into the selected structure (Figure 3-16) and provide parameter input names and NumericUpDown controls for the structure.

4. Modify the parameters of the structure in the form if it is needed and check the final shape of the structure in the form.

5. Click the yes button to insert the structure or the no button to cancel the action.

The top face of the structure, defined by $w_i$ and $l_i$ in Figure 3-16, has a height defined in the form. After the mesh is created and the elevation at each mesh vertex is interpolated, you can use the Mesh>Set-LWD-Elevation to calculate the vertex elevations of the structure to match the surrounding terrain elevation and defined structure height.

## 3.3.28. Insert a More Complex Large Wood Structure

SRH-Mesh enables you to add four types of more complex pre-defined large wood structures (Figure 3-17).
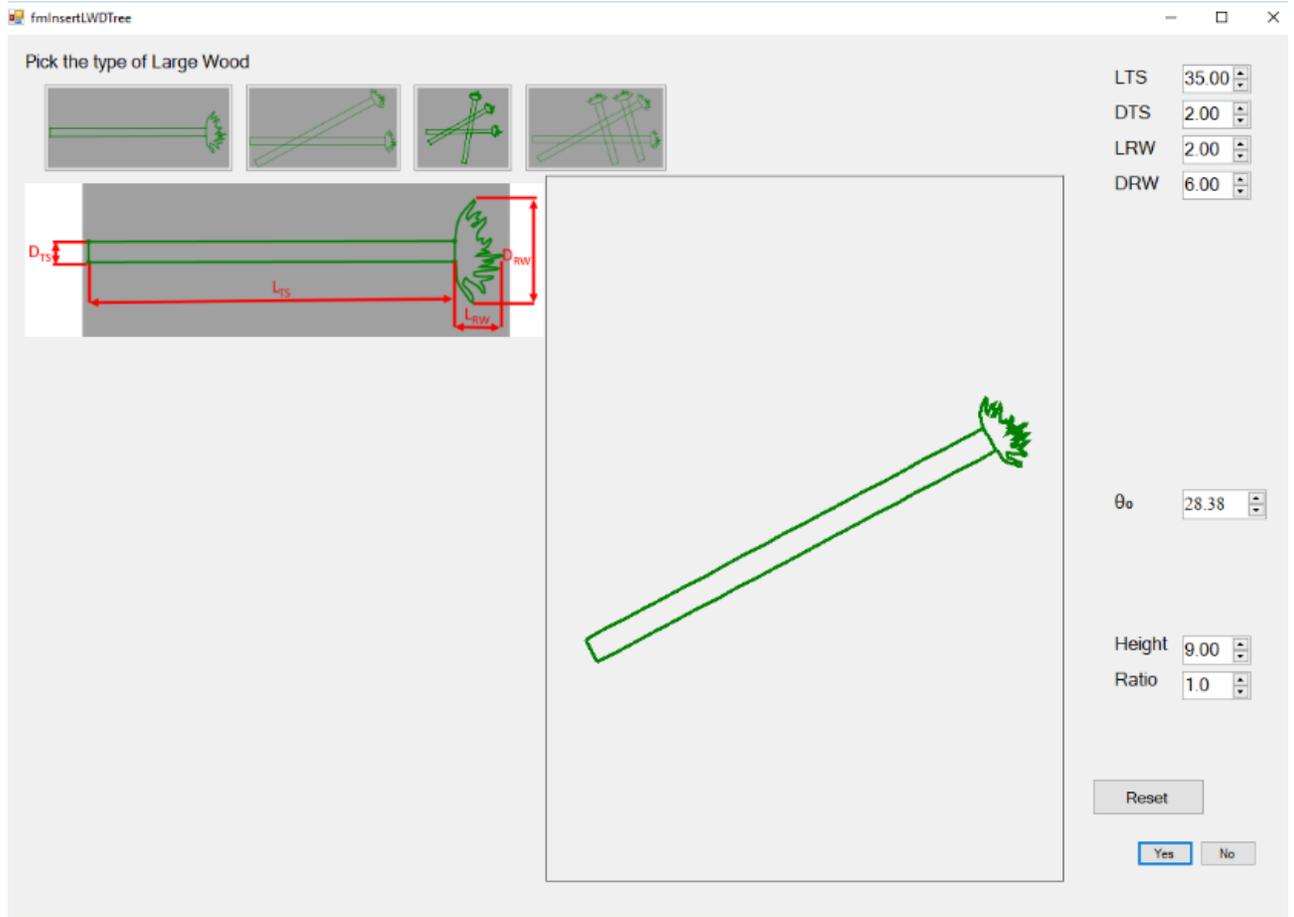


**Figure 3-17. Form to select the type of large wood structure and set the structure's dimensions.**

To add a large wood structure:

1. Select Geometry>Insert Large Wood Structures.

2. Click a point in the workspace where you want to insert the beginning of the structure, hold and drag to a new point where you want to insert the end of the structure. The direction from the first to second point represents the orientation of the large wood structure. The middle point of your mouse "hold and drag" represents the center of the large wood structure. A form will appear (Figure 3-17).
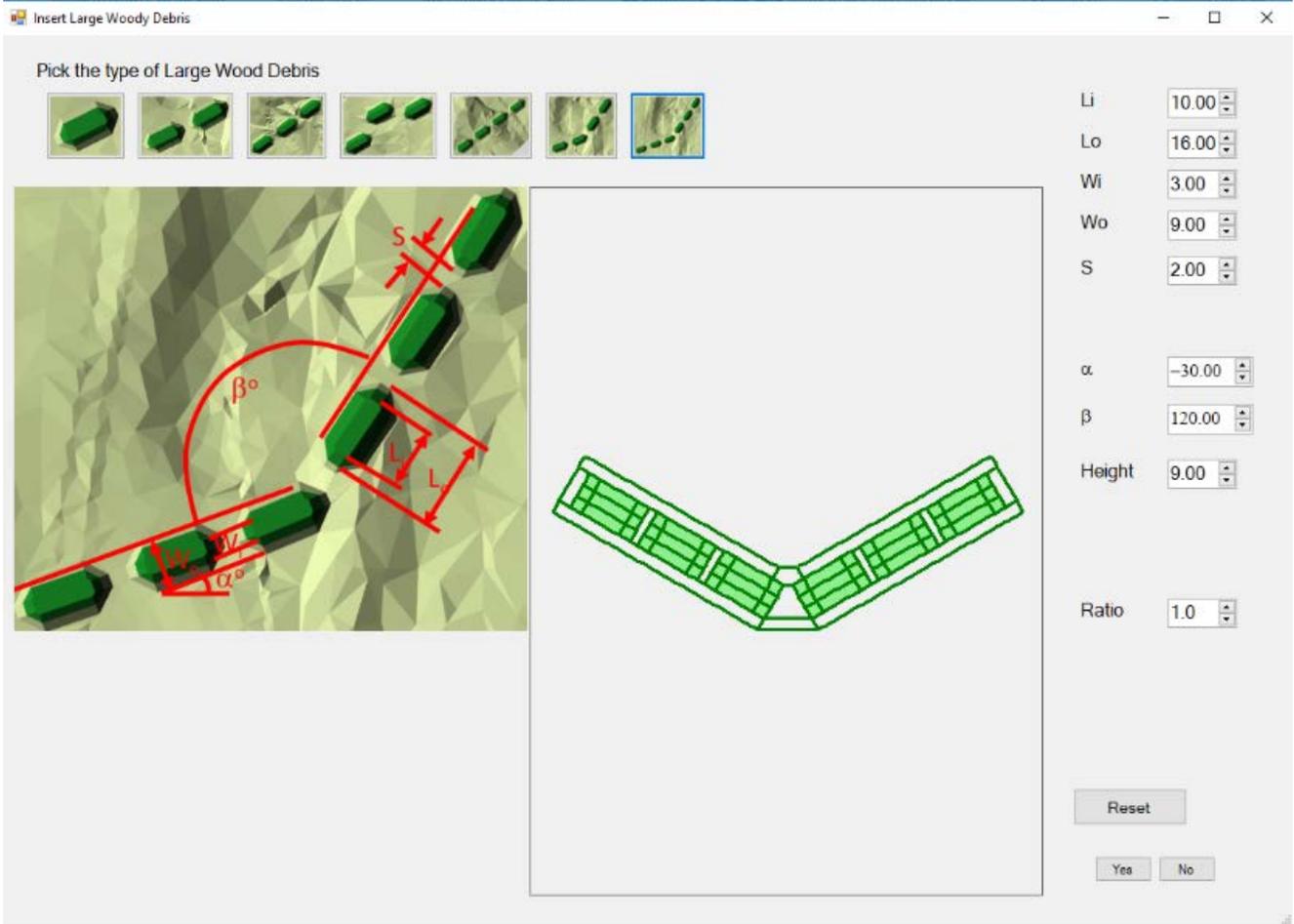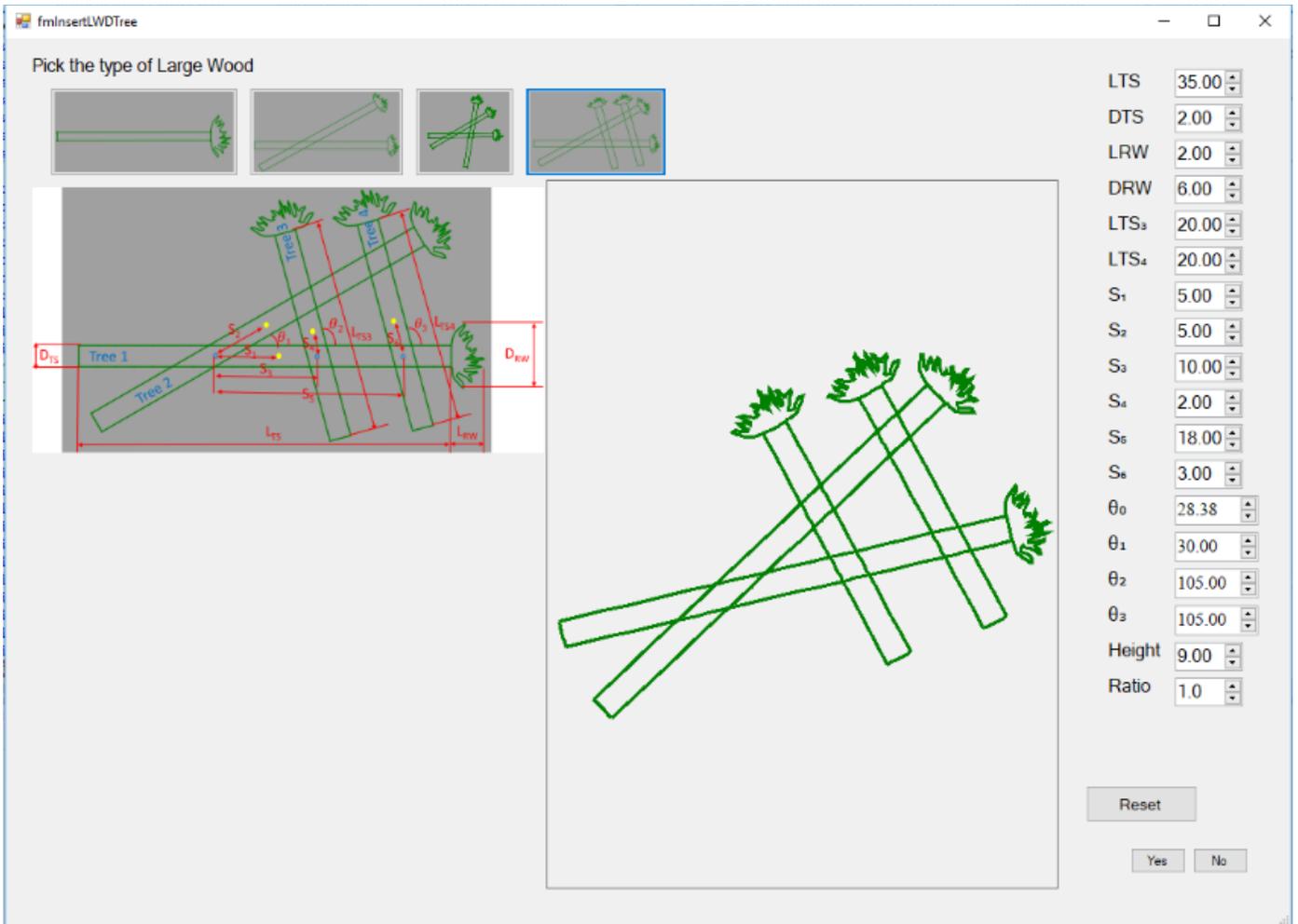
**Figure 3-18. Form to pick the type of large wood structure and set the dimensions of the structure.**

3. Select one of the four types of more complex pre-defined large wood structures in the form. The form will change its representing figure into the selected structure (Figure 3-18) and provide parameter input names and NumericUpDown controls for the structure.

4. Modify the parameters of the structure in the form if it is needed and check the final shape of the structure in the form.

5. Click the yes button to insert the structure or the no button to cancel the action.

## *3.4. Individual and Global Mesh Module*

Click Mesh in the table of contents to switch to Mesh Module toolbar (Figure 3-19). The icon for

the Mesh Module will appear in the lower left corner of the workspace: individual ▯▷ and

global ▷.



**Figure 3-19. Mesh Module toolbar..**

## 3.4.1. Select Outside Line(s) for Mesh

Selected outside lines are displayed in cyan. To create a mesh, make sure that all outside lines are connected to form one or polygons.

1. Click select outside line button

2. Use mouse to select the line(s)

   a. Click to select a line

   b. Press Ctrl + click to add or remove selection(s). Press Alt + click to add selection(s)

   c. Click and drag to select all of the lines in a drag box

## 3.4.2. Select Inside Line(s) for Mesh

Inside lines should not intersect with outside lines or hole lines. Selected inside lines are displayed in yellow. Inside lines are used for triangle meshes, mixed meshes, or Voronoi meshes where each node on an inside line is a vertex in the resulting mesh. Inside lines are ignored by the quadrilateral mesh generator.

1. Click the select inside line button

2. Use mouse to select the line(s)

   a. Click to select a line

   b. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s)

   c. Click and drag to select all lines in the drag box

## 3.4.3. Select Hole Line(s) for Mesh

Hole lines should not intersect with outside lines or inside lines. Selected hole lines are displayed in brown. Hole lines are used for triangle meshes, mixed meshes, or Voronoi meshes where holes are present. Hole lines are ignored by quadrilateral mesh generator.

To create a mesh, make sure that all hole lines are connected to form one or more polygons.

1. Click the select hole line button ![icon]

2. Use mouse to select the line(s).

   a. Click to select a line

   b. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s)

   c. Click and drag to select all of the lines in a drag box

## 3.4.4. Create a Triangle Mesh

1. Select outside lines (Section 3.4.1), inside lines (optional) (Section 3.4.2), and hole lines (optional) (Section 3.4.3).

2. Make sure that all outside lines are connected to form one or more polygons.

3. Make sure that all hole lines are connected to present one or more polygons.

4. Make sure that inside lines do not have any intersections with outside lines and hole lines.

5. Click the triangle-mesh button ![icon].

6. Check the quality of the mesh. To improve the quality of the mesh, right click line(s) for the mesh, and a form for line redistribution appears. Refer to Section 3.3.7 for details of line redistribution.

## 3.4.5. Create a Quadrilateral Mesh

1. Select outside lines (Section 3.4.1).

2. Make sure that all outside lines are connected to form one or more polygons.

3. Click the quadrilateral-mesh button ![icon].

4. Check the quality of the mesh. To improve the quality of the mesh, right click line(s) for the mesh, and a form for line redistribution appears. Refer to Section 3.3.7 for details of line redistribution.

5. Check the sides of the created quadrilateral mesh. If there are more than four lines selected as a boundary for a mesh, SRH-Mesh will automatically detect the shape of the boundary and determine how to group several lines into a side of the mesh. If you want to use another way to composite a side of a mesh, you can group these connected lines by setting a same group number for the connected lines. Please refer to grouping lines (Section 3.3.19) and ungrouping lines (Section 3.3.20).

## 3.4.6. Create a Triangle and Quadrilateral Mixed Mesh

1. Select outside lines (Section 3.4.1), inside lines (optional) (Section 3.4.2), and hole lines (optional) (Section 3.4.3).

2. Make sure that all outside lines are connected to form one or more polygons.

3. Make sure that all hole lines are connected to form one or more polygons.

4. Make sure that inside lines do not have any intersections with outside lines and hole lines.

5. Click the mixed-mesh ▲ button.

6. Check the quality of the mesh. To improve the quality of the mesh, right click line(s) for the mesh, and a form for line redistribution appears. Refer to Section 3.3.7 for details of line redistribution.

## 3.4.7. Create a Voronoi Mesh

If one mesh is created as a Voronoi mesh, all meshes should be created as Voronoi.

1. Select outside lines (Section 3.4.1), inside lines (optional) (Section 3.4.2), and hole lines (optional) (Section 3.4.3).

2. Make sure that all outside lines are connected to form one or more enclosed mesh boundaries.

3. Make sure that all hole lines are connected to form one or more enclosed holes.

4. Click the Voronoi-mesh ⬠ button.

5. Check the quality of the mesh. To improve the quality of the mesh, right click line(s) for the mesh, and a form for line redistribution appears. Refer to Section 3.3.7 for details of line redistribution.

## 3.4.8. Select Mesh(es)

A mesh or multiple meshes are selected for several purposes: to change the type of the mesh, to remove the mesh, to set the bed roughness of the mesh, to set the initial condition zone of the mesh, or to set the sediment gradation zone of the mesh.

1. Make sure that the mesh display mode is of individual mesh. Individual meshes are displayed in two places: the label for individual mesh controls is "MHI" and the module indicator is ⬛. Refer to Sections 3.4.13 and 3.4.14 for details of individual mesh and global mesh modes.

2. Click the select mesh button ⬛.

3. Click on a mesh to select it.

   a. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s).

   b. Click and drag to select all of the meshes in a drag box.

## 3.4.9. Select Mesh Cell(s)

Mesh cells are selected for several purposes: to set the bed roughness of the cell(s), to set the initial condition zone of the mesh cell (s), or to set the sediment gradation zone of the mesh cell(s). Mesh cell (s) can be selected in both individual mesh and global mesh modes.

1. Click the select mesh cell button 

2. Use a mouse to select the mesh face(s)

   a. Click to select a mesh face

   b. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s)

   c. Click and drag to select all of the mesh cells in a drag box

## 3.4.10. Select Mesh Vertex and Set Elevation

A mesh vertex or multiple mesh vertexes are selected only to set the vertex elevation. The mesh elevation is changed on selected mesh vertexes in each individual mesh, and the global mesh elevation will be updated only when the refresh button is clicked.

1. Click the select vertex button 

2. Use mouse to select the mesh face(s)

   a. Click to select a mesh face

   b. Press Ctrl + click to add or remove selection(s) and press Alt + click to add selection(s)

   c. Click and drag to select all of the meshes in a drag box

3. Input the new elevation value in the z textbox in the status line, and press the Enter key to set the elevation

## 3.4.11. Deselect All

Click the deselect all button  to deselect all in the geometry, mesh, and boundary condition modules.

## 3.4.12. Delete Selected Mesh

1. Select the mesh(es) to be deleted

2. Click the delete mesh button 

## 3.4.13. Switch to Individual Mesh Mode

The Individual Mesh mode shows the boundary lines of each mesh. Several mesh functions are only available in individual mesh mode (e.g., select the mesh, change mesh type, change mesh roughness, change mesh initial condition zone, and change mesh bed gradation zone).

You can switch to the individual mesh mode either by clicking the individual meshes button  or on the MHI label in the mesh controls.

## 3.4.14. Switch to Global Mesh Mode

Global mesh mode shows the final mesh after joining all individual meshes together. The boundary lines of each individual mesh are not displayed and only boundary lines of the joined mesh are displayed. 2D models, such as the SRH-2D, can only use the global mesh mode.

You can switch to the global mesh mode either by clicking the global mesh button or on the MHG label in the mesh controls.

## 3.4.15. Change the Bed Roughness Zone in Selected Mesh(s)

Make sure that bed roughness is displayed (Section 3.4.22), and individual mesh mode is selected (Section 3.4.8).

1. Refer to Section 3.4.9 to select the mesh(es) of which bed roughness zone will be set.

2. Click the change mesh bed roughness zone button . A form will appear.

3. Select the bed roughness zone.

## 3.4.16. Change the Bed Roughness Zone in Selected Mesh Cell(s)

Make sure that bed roughness is displayed (Section 3.4.22).

1. Refer to Section 3.4.9 to select the mesh cell(s) that you want to set the bed roughness zone for.

2. Click the change cell bed roughness zone button . A form will appear.

3. Select the bed roughness zone.

## 3.4.17. Change the Sediment Gradation Zone in Selected Mesh(s)

The bed gradation zone is only available when the Morphology solver is selected and the sediment gradation is set as zone input. Make sure that sediment gradation zone is displayed (Section 3.4.22), and individual mesh mode is selected (Section 3.4.13).

1. Select the mesh(s) (Section 3.4.9) that you want to set the sediment gradation zone for.

2. Click the change mesh sediment gradation zone button . A form will appear.

3. Select the sediment gradation zone.

## 3.4.18. Change the Sediment Gradation Zone in Selected Mesh Cell(s)

The sediment gradation zone is only available when the Morphology solver is selected, and the sediment gradation is set as zone input. Make sure that sediment gradation zone is displayed (Section 3.4.22).

1. Select the mesh cell(s) (Section 3.4.9) that you want to set the sediment gradation zone for.

2. Click the change cell sediment gradation zone button . A form will appear.

3. Select the sediment gradation zone.

## 3.4.19. Change the Initial Condition Zone in Selected Mesh(s)

The initial condition zone is only available when the Flow Unsteady or Morphology solver is selected, and the initial condition is set as zone input. Make sure that the initial condition zone is displayed (Section 3.4.22), and that the individual mesh mode is selected (Section 3.4.13).

1. Select the mesh(s) (Section 3.4.9) that you want to set the initial condition zone for.

2. Click change mesh initial zone button . A form will appear.

3. Select the initial condition zone.

## 3.4.20. Change the Initial Condition Zone in Selected Mesh Cell (s)

The initial condition zone is only available when the Flow Unsteady or Morphology solver is selected, and the initial condition is set as zone input. Make sure that bed roughness is displayed (Section 3.4.22).

1. Select the mesh(s) (Section 3.4.9) that you want to set the initial condition zone for.

2. Click the change cell initial zone button . A form will appear.

3. Select the initial condition zone.

## 3.4.21. Display Mesh Information

Click the mesh information button  to display mesh information (Figure 3-20). If the individual mesh mode is displayed and only one mesh is selected, then information about the individual mesh is displayed. If the global mesh mode is displayed or two or more meshes are selected, then global mesh information is displayed.
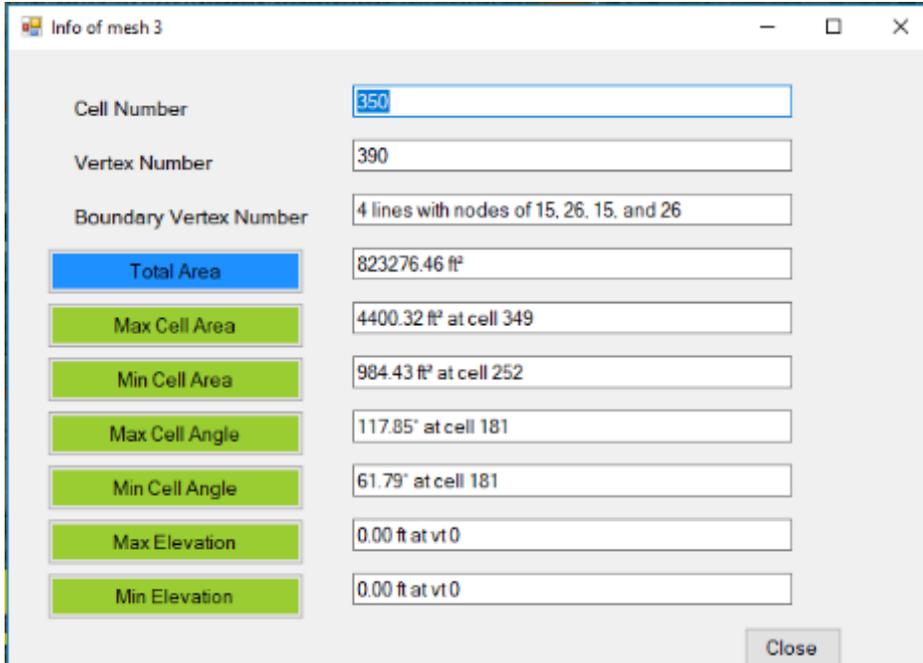
**Figure 3-20. Mesh information.**

This information is shown:

- **Cell Number:** the total number of cells.

- **Vertex Number:** the total number of vertexes.

- **Boundary Vertex Number:** the total number of lines in the boundary and number of vertexes associated with each line are displayed for an individual mesh. The number of vertexes is displayed for global mesh.

- **Total Area:** the area of the selected mesh. You can right-click the total area button to switch the units between square feet (ft$^2$), square meters (m$^2$), and acres.

- **Max Cell Area:** the area and index of the cell with the largest (maximum) area. Click the max cell area button to switch the workspace zoom between the mesh and the cell with the maximum area. Double-click the max cell area button to zoom the workspace into the cell with maximum area. Right-click the button Max Cell Area to switch the units between ft$^2$, m$^2$, and acres.

- **Min Cell Area:** the area and index of the cell with a minimum area. Click the min cell area button to switch the workspace zoom between the mesh and the cell with minimum area. Double-click the min cell area button to zoom the workspace into the cell with minimum area. Right-click min cell area button to switch the units between ft$^2$, m$^2$, and acres.

- **Max Cell Angle:** the degree of the maximum angle and cell index of the cell with that maximum angle. Click the max cell angle button to switch the workspace zoom between the mesh and the cell with maximum angle. Double-click the max

cell angle button to zoom the workspace into the cell with maximum angle.

- **Min Cell Angle:** the degree of the minimum angle and cell index of the cell with a minimum angle. Click the min cell angle button to switch the workspace zoom between the mesh and the cell with minimum angle. Double-click the min cell angle to zoom the workspace into the cell with minimum angle.

- **Max Elevation:** the elevation and index of the vertex with a maximum elevation. Click the max elevation button to zoom the workspace into the vertex with maximum elevation.

Click the close button to close the form.

## 3.4.22. Set Mesh Display Options

1. Click the set display button . A form will appear (Figure 3-21).

2. You can adjust these input parameters:

   a. Line style, width, and color.

   b. Node style, size, and color.

   c. Domain boundary line style, width, and color.

   d. Node index label font and color.

   e. Mesh index label font and color.

   f. Cell index label font and color.

   g. Vertex elevation map option.

   h. Roughness zone name, forecolor, background color, transparency level, and fill pattern. Right-click the mouse on the table to add a zone or remove a zone.

Figure 3-21. Mesh format.

**Note:** The bed gradation zone is only available when the Morphology solver is selected and the sediment gradation is set as zone input. You can adjust the bed gradation zone name, forecolor, background color, transparency level, and fill pattern. Right-click the mouse on the table to add a zone or remove a zone.

The initial condition zone is only available when the Flow Unsteady or Morphology solver is selected, and the initial condition is set as zone input. You can adjust the initial condition zone name, forecolor, background color, transparency level, fill pattern. Right-click the mouse on the table to add a zone or remove a zone.

3. Click the yes button to save the mesh display options or the no button to cancel the action.

## 3.4.23. Refresh Mesh

Elevation changes are made only to vertexes in each individual mesh, and the global mesh elevation will be updated only when the refresh button is clicked.

Click the refresh mesh button ⤸ to refresh the mesh.

## 3.4.24. Save Mesh

Click the save created mesh button 💾 to save the mesh in the ".srhm" format.

## 3.4.25. Refine Mesh

The refine mesh tool is used only when all meshes are created and you want to change the density of the final mesh in a single step. This tool will refine geometry lines (outside lines, inside lines, and hole lines) associated with all individual meshes, and refine all meshes accordingly.

1. Select Menu Mesh>Refine Mesh. A form will appear (Figure 3-22).
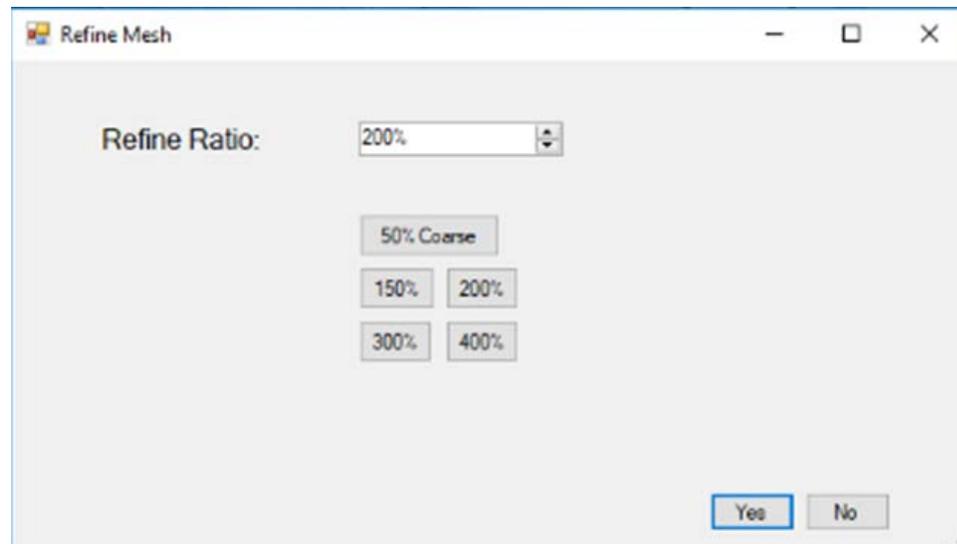


Figure 3-22. Refine mesh.

2. Input the refine ratio. The ratio should be in the range of 50% (where the space between original vertexes is doubled) to 400% (where the space between two original vertexes is divided into 4 segments). Quick buttons of 50%, 150%, 200%, and 400% are provided.

3. Click the yes button to perform the action and the no button to cancel the action.

## 3.4.26. Assign the Roughness Zone with GIS Shapefile

This tool will assign all mesh cells according to the field selected in the database associated with input GIS shapefile. The database file "***.dba" with the same name as the shapefile and extension ".dba" is required in the same folder. The GIS shapefile should be a polygon (or polygonZ/polygonZM) type. The GIS Shapefile should have the same projection with the SRH-Mesh project.

1. Select Mesh>Assign Roughness Zone with GIS Shapefile. A form will appear (Figure 3-23).
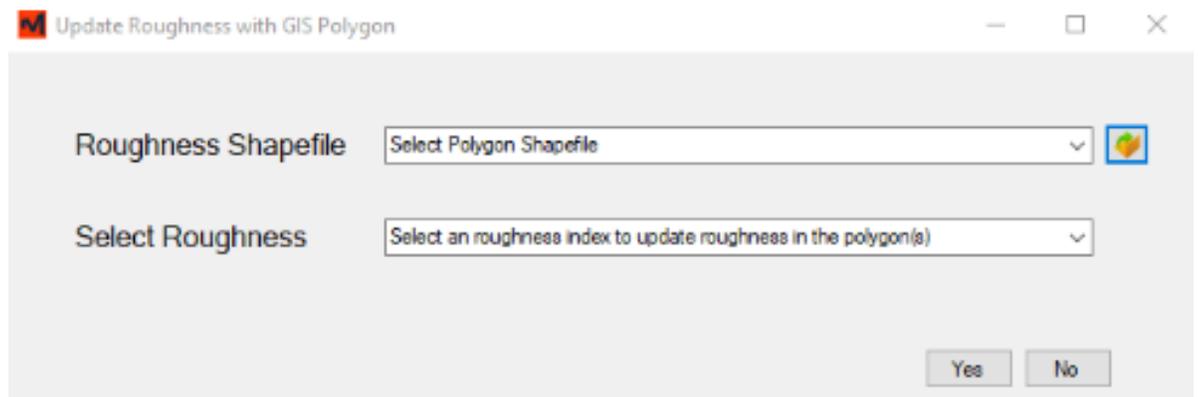


**Figure 3-23. Assign roughness with GIS polygon field.**

2. Select parameters:

   a. **Roughness Shapefile:** select the GIS shapefile used to assign roughness from the drop-down menu or from an open file form

   b. **Select Roughness:** select the roughness index used for all cells within the GIS shapefile polygons from a drop-down menu.

3. Click the yes button to perform the action and the no button to cancel the action

## 3.4.27. Update Roughness Zone with GIS Shapefile

This tool will update the roughness zone number in cells only inside the polygons which are defined in an input GIS shapefile to a user-defined value. The GIS shapefile should be a polygon (or polygonZ/polygonZM) type. The GIS shapefile should have the same projection with the SRH-Mesh project.

1. Select Mesh>Update Roughness Zone with GIS Shapefile. A form will appear (Figure 3-24)
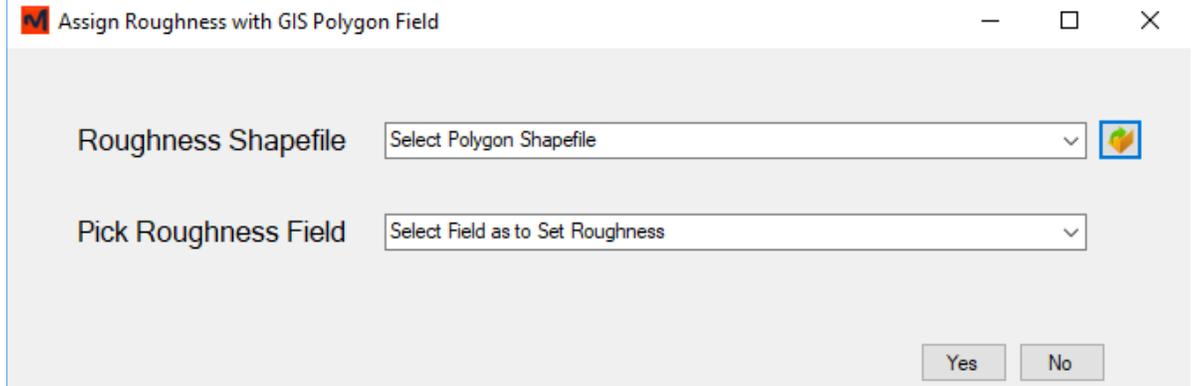
**Figure 3-24. Update roughness with a GIS shapefile.**

2. Input parameters:

   a. **Roughness Shapefile:** Select the GIS shapefile used to update roughness index

   b. **Select Roughness:** Select the GIS field used to update roughness index

3. Click the yes button to perform the action and the no button to cancel the action

## 3.4.28. Interpolate Mesh Elevations to Points in a Text File

1. Select Mesh>Interpolate Mesh Elevations to Points in a File.

2. Select a text file that contains survey points. The file should contain point data for x, y, and z coordinates, separated by space or comma. The file should also include a heading with column names. The heading should be included in A form will appear (Figure 3-25).
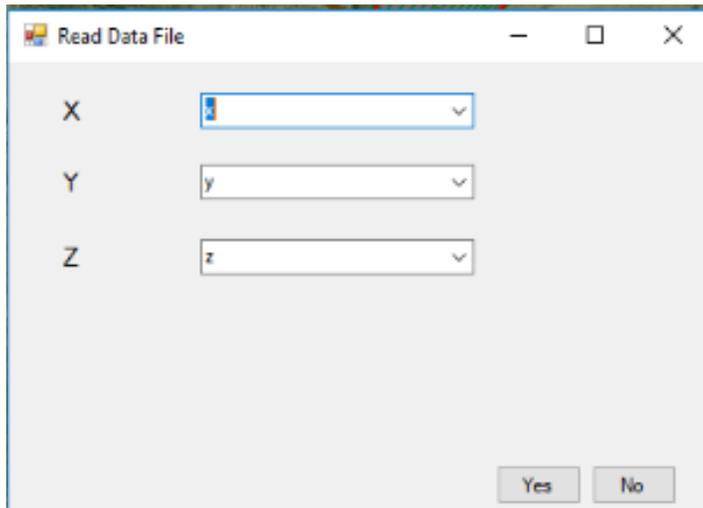


**Figure 3-25. Interpolate mesh elevations to points in a file.**

3. Select columns for x, y, and z coordinates.

4. Click the yes button to perform the action and the no button to cancel the action.

### 3.4.29. Interpolate Mesh Elevations to GIS Point Shapefile

1. Select Mesh>Interpolate Mesh Elevations to Survey Points in Shapefile bounded by Polygon in Shapefile. A form will appear (Figure 3-26).
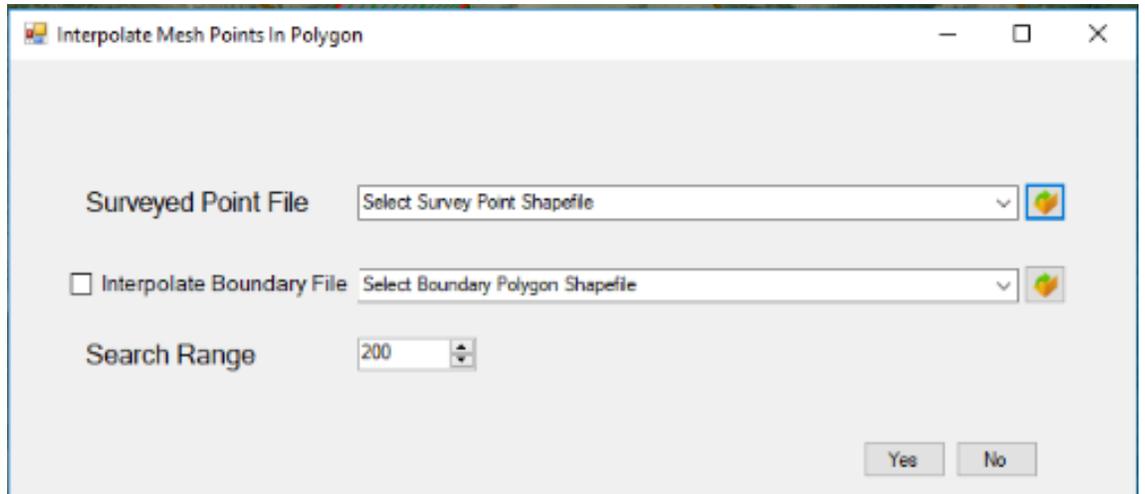


**Figure 3-26. Interpolate mesh elevations to GIS point shapefile.**

2. Input the data in the form:

   a. **Surveyed Point File:** the GIS point shapefile representing the survey points with x, y, and z coordinates from a drop-down menu or an open-file form.

   b. **Interpolate Boundary File:** the GIS polygon shapefile representing boundary outside of which surveyed points will not be used to interpolate the mesh vertex elevations.

   c. **Search Range:** the vertex elevation will only be interpolated from survey points whose distance from the vertex is less than the search range.

3. Click the yes button to perform the action and the no button to cancel the action.

## 3.4.30. Interpolate Mesh Elevations to GIS Triangulated Irregular Network (TIN) File

1. Select Mesh>Interpolate Mesh Elevations to Survey Points to GIS TIN. A form will appear (Figure 3-27).
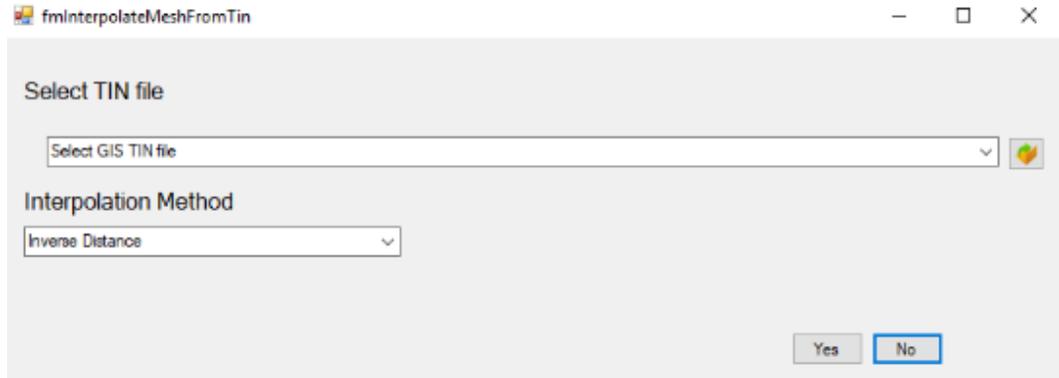


**Figure 3-27. Interpolate mesh elevations to GIS TIN.**

2. Input the data in the form:

   a. **GIS TIN file**: the GIS TIN file with elevation information from a drop-down menu or an open-file form.

   b. **Interpolation Method:** the method used to interpolate the elevation.

3. Click the yes button to perform the action and the no button to cancel the action.

## 3.4.31. Create Mesh from all Geometry

Select Geometry>Create Mesh with All Geometry Lines or Mesh>Create Mesh with All Geometry Lines.

## 3.4.32. Refine All Meshes

1. Click the edit ✎ button to enter the editor toolbar if editor controls (**Error! Reference source not found.**) are not shown.

2. Click the select button ➤ in the editor toolbar and select the line to be redistributed.

3. Click button Redist, and a form appears (**Error! Reference source not found.**).

**Figure 3-28. Geometry line redistribution form.**

4. Input the parameters:

   a. **Node Number:** the total number of vertexes.

   b. **Last/First Ratio:** the ratio of the last segment length to first segment length.

   c. **Neighbor Ratio:** the ratio of length of segment i+1 to length of segment i.

   d. **Cell Length:** the cell length of each segment. The line will be equally distributed.

   e. **Refine:** The line will be refined according to input ratio.

5. Options to distribute the line(s):

   a. Click the yes button to perform the defined redistribution task or the no button to cancel the action.

   b. Click revise ratio button to recalculate the Last/First or Neighbor Ratio as the reciprocal of the ratio and perform the redistribution task.

   c. Click the revise direction button to redistribute the line according to input parameters, then reverse the direction of the line.

6. Continue to edit geometry lines, or click the save button ![save] to save edited geometry, or click the save and exit button ![save and exit] to save edited geometry and exit the editor  toolbar or click the cancel button ![cancel] to exit editor  toolbar without saving.

## 3.5.  Boundary Condition Line Module

Click BC Line in the table of contents to switch to Boundary Condition Line Module toolbar (Figure 3-29). The icon for the Geometry Module will appear in the lower left

corner of the workspace ⬚.



**Figure 3-29. Command buttons for the Boundary Condition Line Module.**

Note that only the global mesh is displayed inside Boundary Condition Line module.

## 3.5.1. Select a Boundary Condition Line or Monitoring Line

1. Click the select button ⬚

2. Use the mouse to select the mesh(es).

   a. Click to select a line

   b. Use Ctrl + click to add or remove selection(s) and use Alt + click to add selection(s).

   c. Use click and drag to select all lines in the drag box.

Or

Double-click the name of the boundary condition line in the table of contents to select the line.

## 3.5.2. Deselect All

Click the deselect all button ⬚ to deselect all in the geometry, mesh, and boundary condition modules.

## 3.5.3. Add a Boundary Condition Line

1. Click the add BC Line button ⬚

2. Click to a vertex at the boundary of the mesh, and the boundary condition line will be extended to the new added vertex

3. Double click to finish adding a line

4. Input a name for the BC line when prompted

## 3.5.4. Add a Monitoring Line

A boundary condition line is always located on the boundary of the mesh, and a monitoring line can lay on the boundary or inside the mesh. No difference is made between these two types of lines once they are created, so the monitoring line is also called the boundary condition line in SRH-Mesh.

1. Click the add monitoring line button ⬚.

2. Click a vertex at the boundary or inside of the mesh, and the monitoring line will be extended to the new added vertex going through the shortest path.

3. Double click to finish adding a line.

4. Input a name for the monitoring line when prompted.

### 3.5.5. Delete Selected Boundary Condition Line(s) or Monitoring Line(s)

1. Select the boundary condition line(s) or monitoring line(s) to be deleted (Section 3.5.1)

2. Click the remove button ▬

Or

1. Click the boundary condition line name in the table of contents to select the line

2. Click the remove button ▬

### 3.5.6. Add Monitoring Point

1. Click add monitoring point button ✚

2. Click a vertex and a monitoring point appears

### 3.5.7. Delete Monitoring Point

1. Click the delete monitoring point button ✖

2. Click an existing monitoring point and that point will be deleted

### 3.5.8. Display the Boundary Condition Line and Monitoring Line Summary

Click the display boundary condition lines **Σ** button. to display all line information (Figure 3-30). Boundary condition line information (in the first table) includes:

- **BC Line:** the boundary condition line name.

- **Flow BC Type:** the flow type for the boundary condition. If a flow boundary condition type has not been assigned, then "BC Not Assigned" is displayed in red.

- **Sed BC Type:** the sediment type for the boundary condition. If a sediment boundary condition type has not been assigned, then "BC Not Assigned" is displayed in red.

- **BC Index:** the index of boundary condition associated with the line. If a boundary condition has not been assigned to this line, the row is displayed in red.
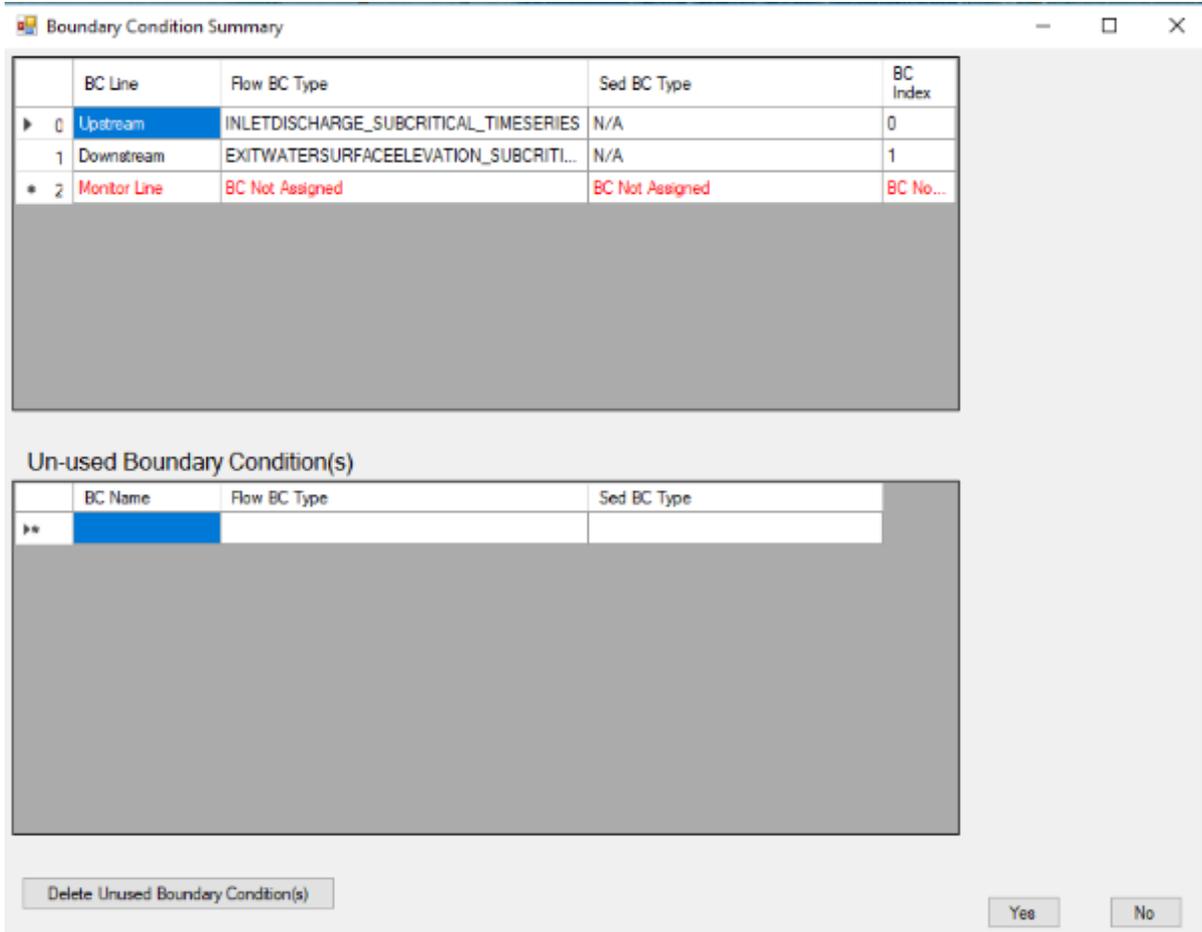
**Figure 3-30. Boundary condition line information.**

The boundary condition depends on boundary condition line. If a boundary condition line is removed, then the associated boundary becomes unused. Unused boundary condition information (in the second table) includes:

- **BC Name:** the boundary condition name
- **Flow BC Type**: the flow boundary condition type
- **Sed BC Type:** the sediment boundary condition type

Click the yes button to close the form or the no button to cancel the action.

## 3.5.9. Display Monitoring Point Summary

Click the monitoring point summary button $\overset{\bullet}{\Sigma}$ to display all line information (Figure 3-31). Monitoring information includes x and y coordinates of all monitoring points.
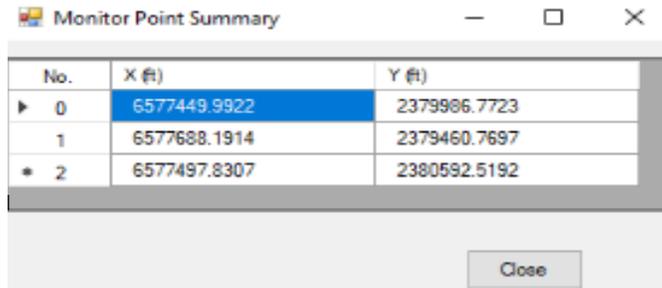


**Figure 3-31. Monitoring point information.**

## 3.5.10.
## Set Boundary Condition Line Display Options

1. Click the set display button . A form will appear (**Error! Reference source not found.**).
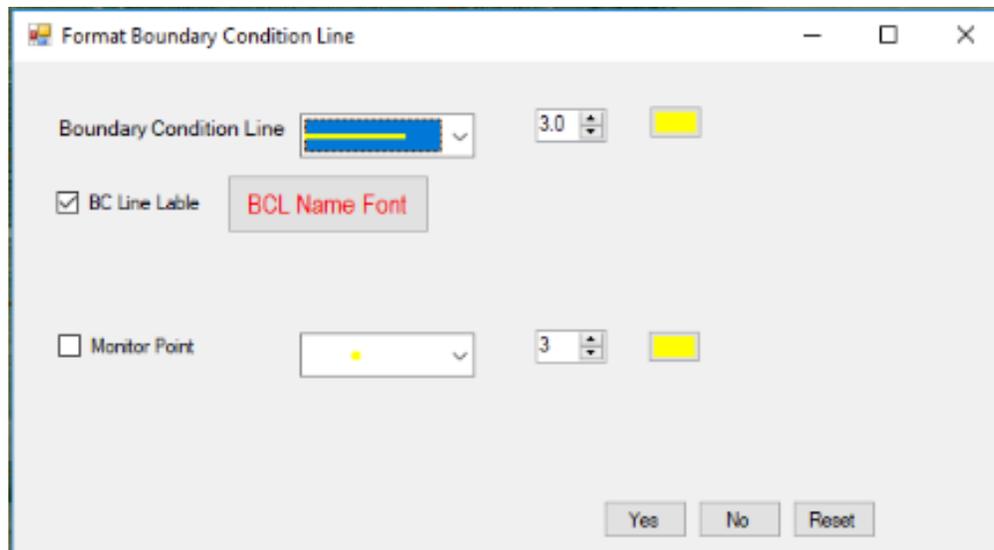


**Figure 3-32 . Boundary condition line format.**

2. Input parameters:

   a. **BC Line:** the style, width, and color for the boundary condition line

   b. **BC Line Label:** the font style, size and color for the boundary condition line label

   c. **Monitor Point:** the style, size and color for the monitoring point

   d. **Reset:** reset to default settings

3. Click the yes button to save the BC line display options or the no button to cancel the action.

## 3.5.11. Save the Boundary Condition Lines

Click the save button 💾 to save boundary condition lines and monitoring points with mesh in ".srhm" format.

## 3.5.12. Edit/Add a Boundary Condition to a Boundary Condition Line

1. Right-click the name of the BC line in the table of contents. A form will appear (Figure 3-33).

2. Select an action from a drop-down menu:

   a. Edit current boundary condition

   b. Add a new boundary condition

   c. Assign an existing boundary condition
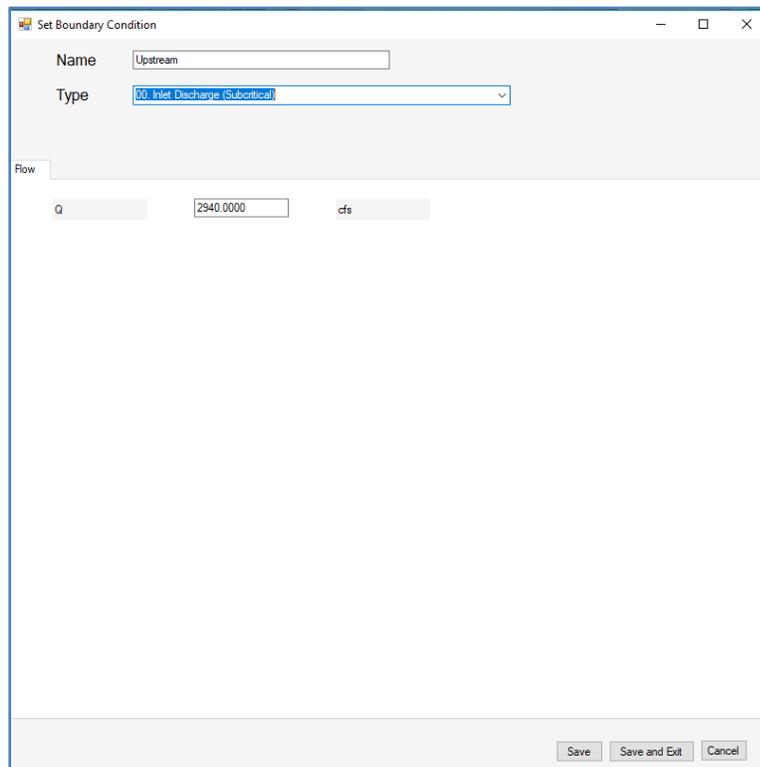


**Figure 3-33. Boundary condition input form.**

3. Input boundary condition parameters:

   a. **Name:** Enter a name for the boundary condition

   b. **Type:** Select flow boundary condition type from drop-down menu

   c. **Flow:** Enter the related flow boundary condition values

## *3.6. Model Setup*

In the Model Setup menu, click Model Setup. A form will appear (Figure 3-34).



**Figure 3-34. Model setup form.**

1. Input the parameters:

   a. **Project Title**

   b. **Mesh Unit:** English or Metric

   c. **Solver:** Steady flow, unsteady flow, or morphology

   d. **Turbulence Model:** Depth-averaged parabolic zero-equation model or standard two-equation k-ε model

   e. **Output Format:** Tecplot, SRHC, or SRHN

2. Click on the Time Step tab and input the parameters (**Error! Reference source not found.**):

   a. Start Time

   b. Time Step

   c. End Time

   d. Output Time Step.
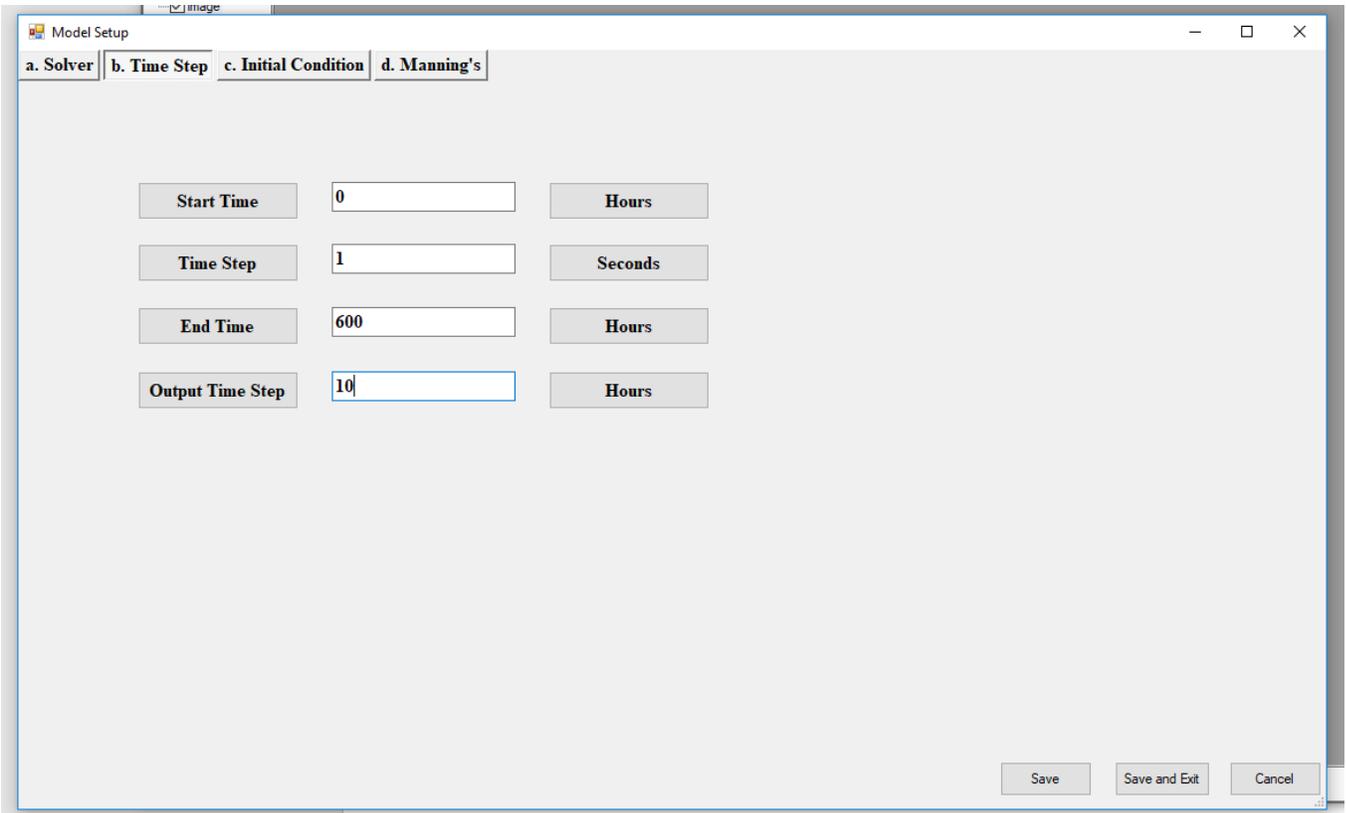
**Figure 3-35. Time step setup in model setup form.**

3. Click on the Initial Condition tab and input the parameters (**Error! Reference source not found.**). Set the Initial Condition to dry, auto, restart, zone, or input.
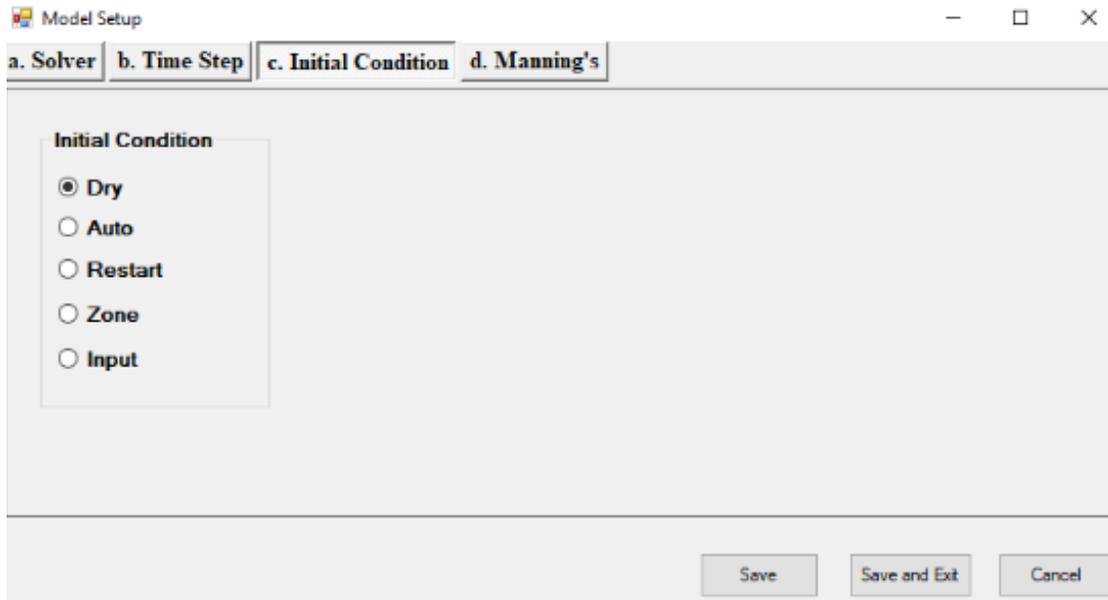


**Figure 3-36. Initial condition setup in model setup form.**

    a.  If "restart" is selected, a restart file is required.

    b.  If "zone" is selected, an initial condition at each zone is required. The initial condition zone is defined in Sections 3.4.19 and 3.4.20.

    c.  If "point" is selected, an initial condition at each point is required. Note that the point initial condition is not available in SRH-2D yet.

4.  Click on the Manning's tab and input the parameters (Figure 3-37). Input Manning's coefficients for each roughness zone. Refer to Section 3.4.22 to define each roughness zone.
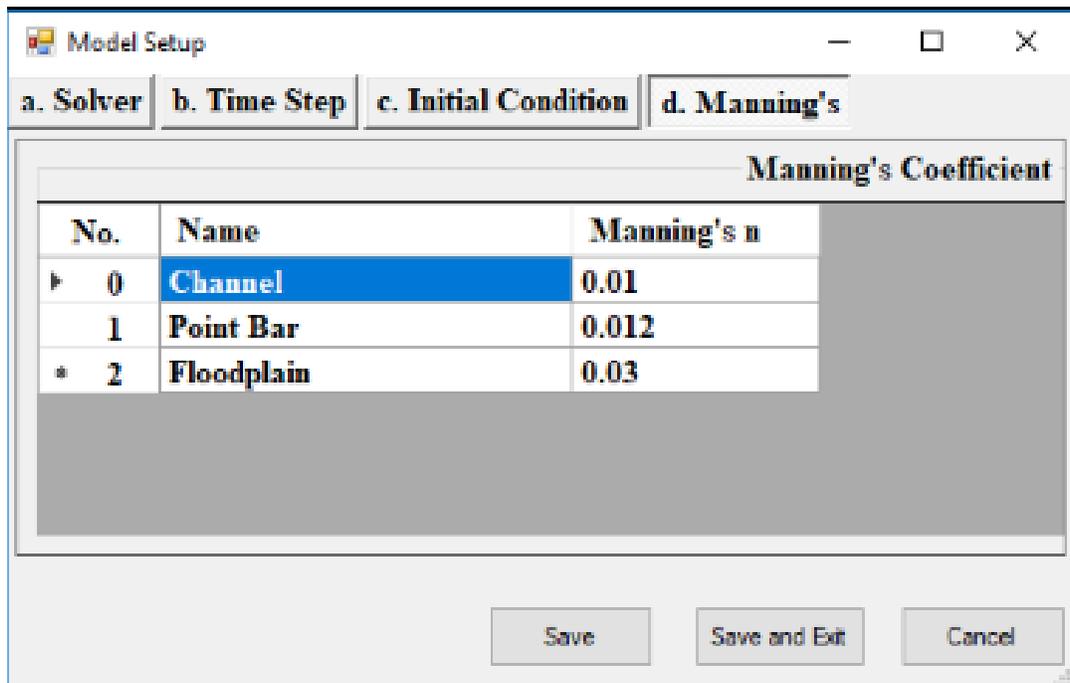


**Figure 3-37. Manning's coefficients setup in model setup form.**

# 4. Tools

This section discusses some of the tools provided in SRH-Mesh menu (Figure 4-1):
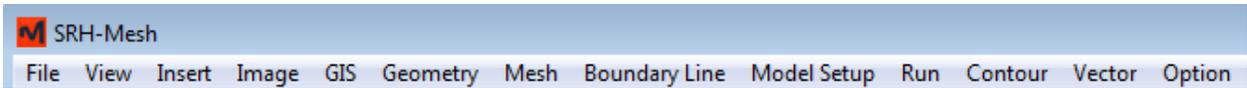


**Figure 4-1. SRH-Mesh's menu bar.**

## 4.1. Find Specific Mesh Node or Cell

In the View menu, click on Find … . A form will appear (Figure 4-2).



**Figure 4-2. Find mesh nodes or cells tool.**

You can find a specific mesh cell or node if you have any of the following information:

- Node with given node number
- Node with given x and y coordinates
- Cell with given cell number
- Cell with given x and y coordinates
- Mesh with given mesh number
- Cell with maximum or minimum cell angles

Select the information type, input the information, and click either Find, Find and exit, or Cancel.

## 4.2. Insert Text

In the Insert menu, click Text and then click a point in the window where you want to insert the text box. A form will appear (Figure 4-3).



**Figure 4-3. Form to insert a text box.**

You can link the text box with GIS coordinates so when the display zoom is changed, the text box is moved to the same location related to GIS, GIS input files, and mesh. If you disable the check-box Location Fixed with GIS Coordinates, then the text box will fit in the window when the display zoom is changed.

You can use this form to change the font style, size, and color with the font button, and you can also change the rotation angle, boundary line, and fill color of the text box in the form.

After a text box is inserted, you can modify it using these options:

- Format the text box. Double-click the text box to bring the same form (Figure 4-3).

- Format, remove, or duplicate the text box. Right-click the text box.

- Size the textbox. Slowly move the mouse at the side midpoint or corner of a text box. After a size handle appears, grab the sizing handle away from or toward the center to increase or decrease the size of the text box.

## *4.3. Insert a Line or a Shape*

In the Insert menu, click Shape and select one of the lines or shapes in the submenu (Figure 4-4), then click a point in the window where you want to insert the line or shape. You can input text inside of any shape. You cannot put text inside or on a line.



**Figure 4-4. Submenu to insert a shape**

You can link a shape with GIS coordinates so that when the display zoom is changed, the shape moves to the same location relative to GIS, GIS input files, and mesh. You can pick one of the nine points (e.g., top left point) in a shape to be fixed to a coordinate point in SRH-Mesh.

If you disable the check-box Location Fixed with GIS Coordinates, the shape will fix in the window when the display zoom is changed.

In the form, you can change the font style, size, and color with the font button of the text inside the shape, and you can also change boundary condition line and fill color of the shape in the form.

After a shape is inserted, you can modify it with these options:

- Format the shape. Double-click the shape to bring a format form (Figure 4-5).

- Format, remove, or duplicate the shape. Right-click the shape.

- Size the shape. Slowly move the mouse at the side midpoint or corner of a shape. After a size handle appears, grab the sizing handle away from or toward the center to increase or decrease the size of the shape.
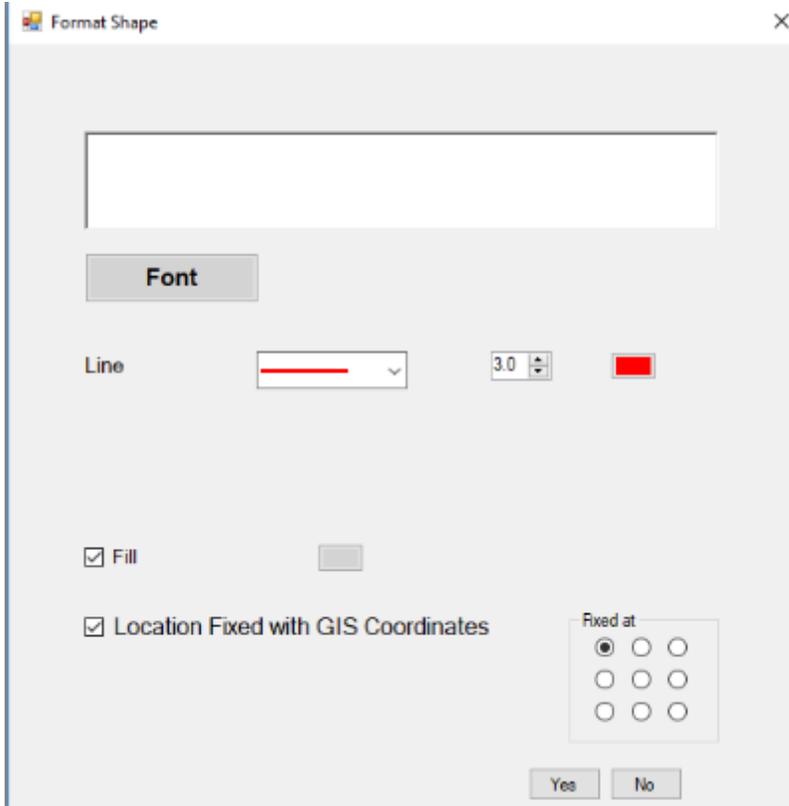
**Figure 4-5. Form to format a shape.**

## 4.4. Insert Marker

In the Insert menu, click Marker and then click a point in the window where you want to insert the marker. A form will appear (Figure 4-6).

Select a type of marker in the combo box and a specific design of the marker in the drop down box at the top of the form. Markers can be pushpin, sphere, square, flag, or place markers—each with different design options.

You can input text at the right side of a marker.

You can use the marker form to change the font style, size, and color with the font button, and you can also change the size and fill color of the marker in the form.

After a marker is inserted, you can modify the marker with these options:

- Format the marker. Double-click the shape to bring a format form ( Figure 4-6).

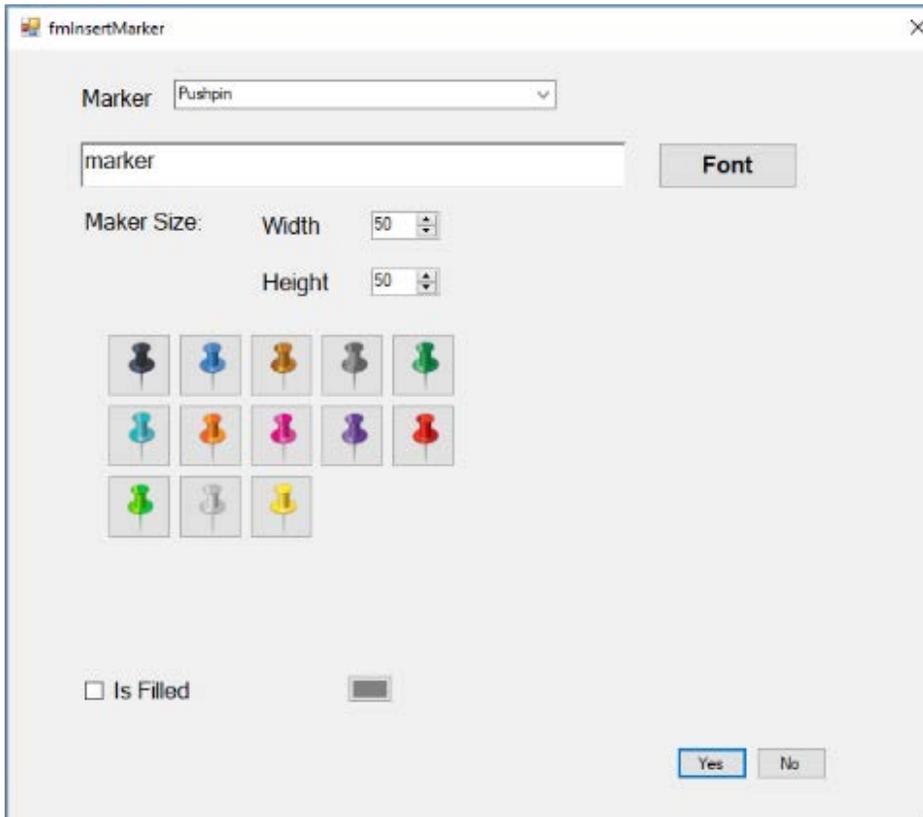- Format, remove, or duplicate the marker. Right-click the marker.

**Figure 4-6. Form to insert a marker.**

Size the shape. Slowly move the mouse at the side midpoint or corner of a shape. After a size handle appears, grab the sizing handle away from or toward the center to increase or decrease the size of the shape.

## 4.5. Insert North Arrow

In the Insert menu, click North Arrow. A form will appear (Figure 4-7).

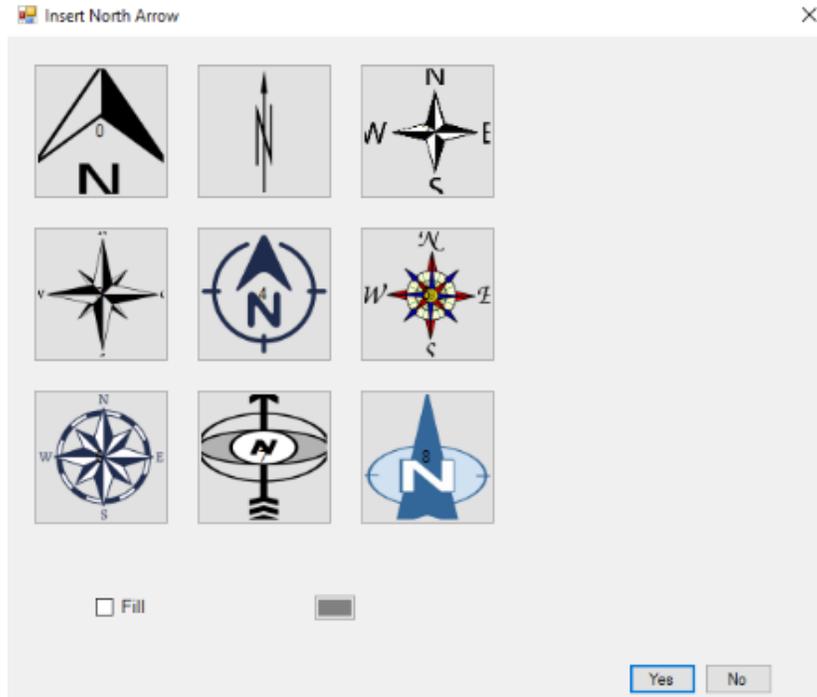Select a specific north arrow in the form, then click the yes button or the no button to cancel the action. A north arrow appears in the workspace.

**Figure 4-7. Form to insert a north arrow.**

After a north arrow is inserted, you can modify it with these options:

1. Move the north arrow to another place. Click the north arrow, hold it, and move it.

2. Format the north arrow. Double-click the north arrow to bring the format form (Figure 4-7). You can change the type of fill color for the north arrow here.

3. Format or remove the north arrow. Right-click it.

4. Size the north arrow. Slowly move the mouse at the side midpoint or corner of the North Arrow. After a size handle appears, grab the sizing handle away from or toward the center to increase or decrease the size of the north arrow.

## 4.6. Insert Scale Bar

In the Insert menu, click Scale Bar. A scale bar appears in the workspace.

After a scale bar is inserted, you can modify it with these options:

1. Move the scale bar to another place. Click the bar, hold it, and drag it.

2. Format the scale bar. Double-click the scale bar to bring the format form (**Error! Reference source not found.**). You can change the type of the combination of the two colors of the scale bar here. You can also change the unit, size, font, border line width and color, and the fill color of the scale bar here.
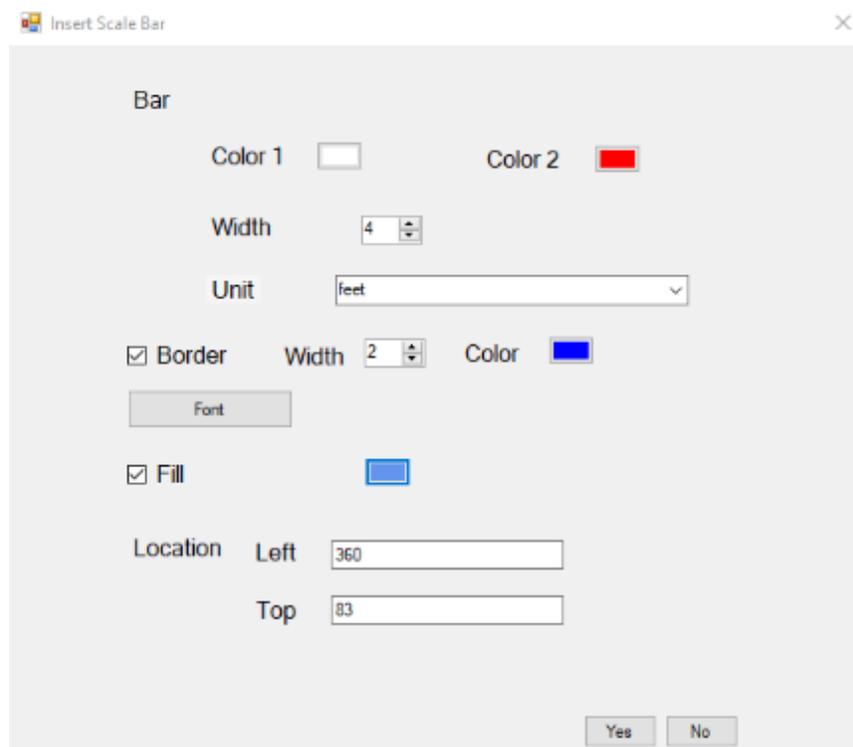
**Figure 4-8. Form to insert a scale bar.**

3.  Format or remove the scale bar. Right click the scale bar.

4.  Size the scale bar. Slowly move the mouse at the side midpoint or corner of the scale bar. After a size handle appears, grab the sizing handle away from or toward the center to increase or decrease the size of the scale bar.

## 4.7. Directional Interpolation

Directional interpolation first calculates the streamwise and transversal coordinates for each survey point by projecting each point into channel center lines. The tool interpolates points in the streamwise and transversal coordinates with a resolution given in the input form (Figure 4-9). The streamwise coordinate is the streamline distance of the projected points to the most upstream point, and the transversal coordinate is the distance from the survey point to the projected point. Directional interpolation uses the nearest point from each quadrant in a curvilinear coordinate plane and interpolates the elevation by using the inverse weighting method.

**Figure 4-9. Form to set up directional interpolation.**

To set up the directional interpolation task:

1. In the Tools menu, click Directional Interpolation. A form will appear (Figure 4-9).

5. Click the ♦ in the line for Channel Center Line File and select a GIS polyline Shapefile (*.shp) as channel center line. Survey points will be projected into the channel center line to calculate the streamwise and transversal coordinates.

6. Click the ♦ in the line for Survey Point File and select a GIS point Shapefile (*.shp) as original survey points. The survey points should include location (x and y) coordinates and an elevation (z) coordinate.

7. Click the ♦ in the line for Interpolate Boundary File and select a GIS polygon Shapefile (*.shp). Only points inside the boundary will be used for interpolation.

8. Input data for streamwise weighting ratio. A larger weighting ratio will put more effect in the streamwise distance.

9. Input data for transverse weighting ratio. A larger weighting ratio will put more effect in the transverse distance.

10. Input data for search range. Only points in a search range will be used for interpolation.

11. Input data for Output Resolution, which is the spacing between interpolated points in both streamwise and transverse directions.

12. Click the yes button to perform the directional interpolation or the no button to cancel the action. Currently, the interpolated points are saved in file "output.dat".

## 4.8. Create GIS TIN from SRH2D Results

A GIS triangulated irregular network (TIN) file can be created from SRH-2D output file in Tecplot format. Follow these steps to generate a TIN.

1. In the Tools menu, select Create GIS TIN from SRH2D Results. A form will appear (Figure 4-10).



**Figure 4-10. form to create a TIN file.**

2. Click the open file button in the line for the Result File and select a SRH-2D result file in Tecplot format.

3. Select the result zone in the selection box.

4. Select the variable in the selection box.

5. Check/uncheck the Include Blanking button if you want to blank cells. If you do not want to blank any cell, go to step 7.

6. To include value blanking, click the add button and select the variable, condition operator ($<$ or $>$) and input the value for the value blanking. You can use more than one conditional statement for value blanking. In the example in Figure 4-10, cells with variable $z>2000$ or $z<1000$ will be trimmed from the final TIN surface.

7. Click the open file button ⬙ in the line for "Folder" and select a folder to save the TIN file.

8. Input the TIN name in the Name textbox.

9. Click the yes button to create the TIN file or the no button to cancel the action.

# 5. Tutorials

This tutorial is based on the Sacramento River.

## *5.1. Tutorial One – Import GIS Files*

Tutorial one shows how to input GIS data such as shapefiles. Generally, the first task of mesh generation is to import GIS data.

Step 1: Launch SRH-Mesh. The start page is shown in Figure 5-1.



**Figure 5-1. SRH-Mesh start page.**

Step 2: SRH-Mesh main page will show up after waiting for 10 seconds. You can also left-click in the start form to switch to SRH-Mesh main page to get the main page to pop up immediately (Figure 5-2).
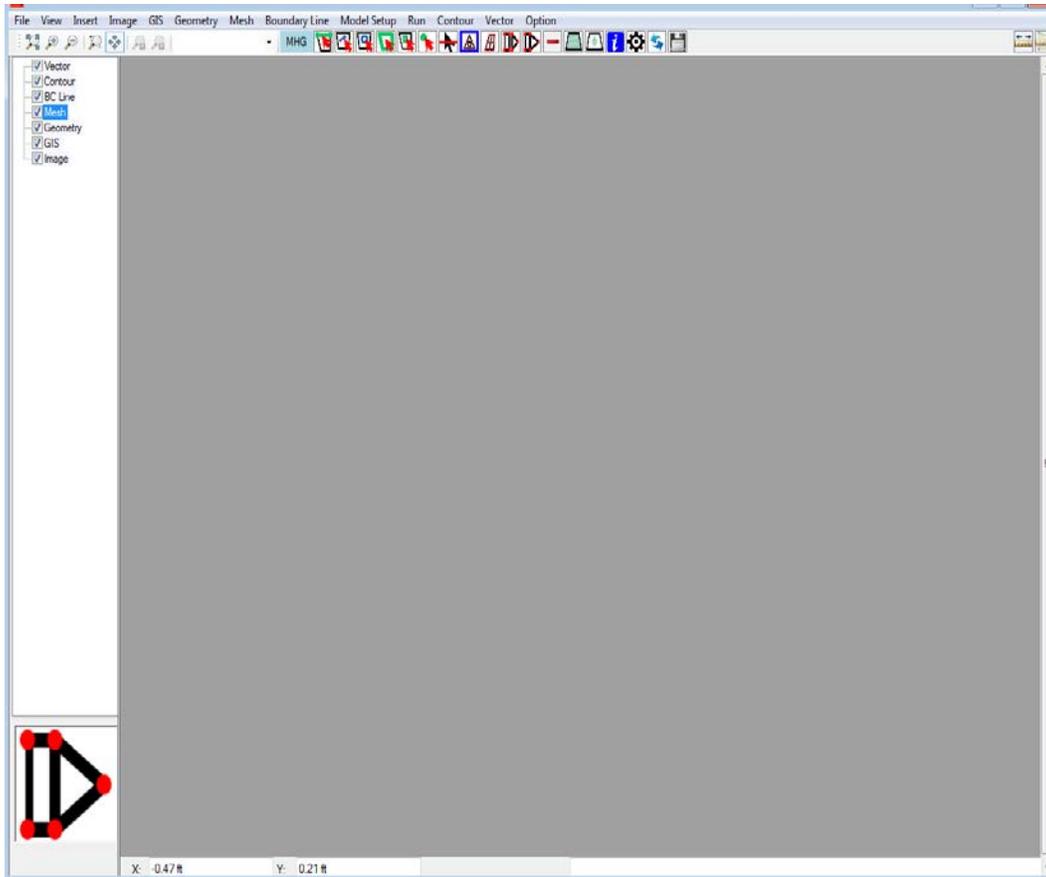
**Figure 5-2. SRH-Mesh main page.**

Step 3: Click GIS in the table of contents and click the add button ➕ in the GIS Module toolbar to import GIS data to the project. For the tutorial case, navigate to the directory where the GIS resides (e.g., GIS Data\Sacramento_RM192) and select "RM192.tif" file to add the map to the project. Use the zoom buttons to check the map (Figure 5-3).
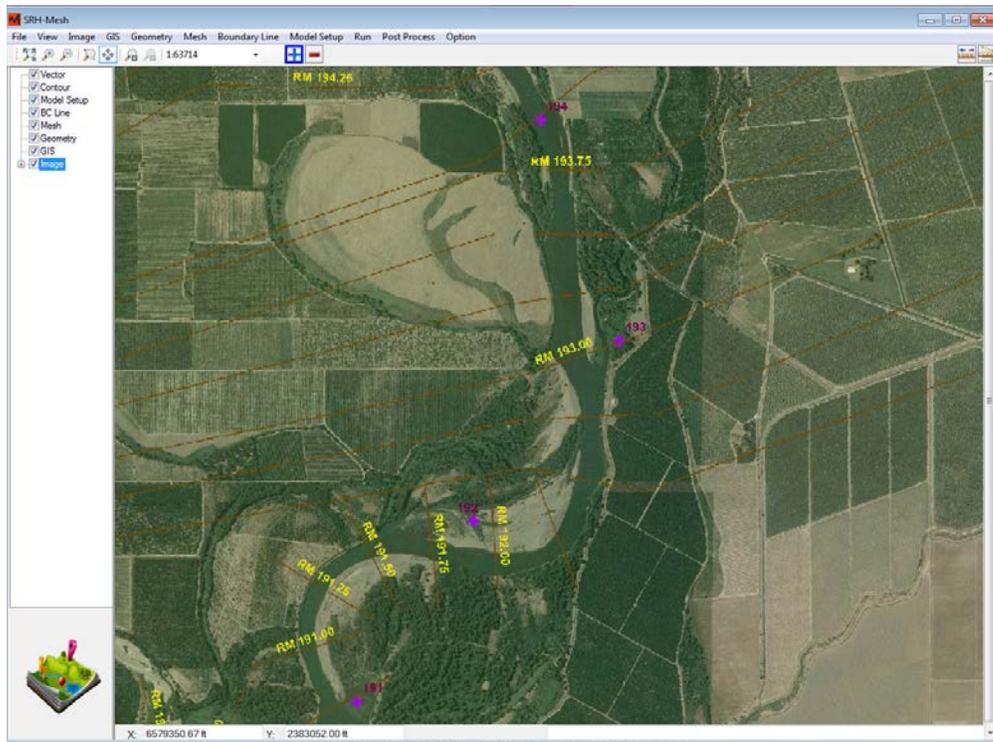
**Figure 5-3. Input Image.**

Step 4: Click the box immediately to the left of the GIS menu item in the table of contents to see the GIS file names. The GIS file named "RM 192.tif" should show up. Check or uncheck the box before GIS or before "RM 192.tif" to show or hide all GIS files or each individual GIS file.

Step 5: Select File\Save Project A and save the project as "tutorial01.shrp".

Step 6: Click the GIS Module in the table of contents and click the add button ✚ in the GIS Module toolbar to add the GIS shapefile to the project. Navigate to GIS file Data\Sacramento_RM192 and select the "boundary.shp" file to add the GIS polyline feature to the project. Use the zoom buttons to check the polyline (Figure 5-4).
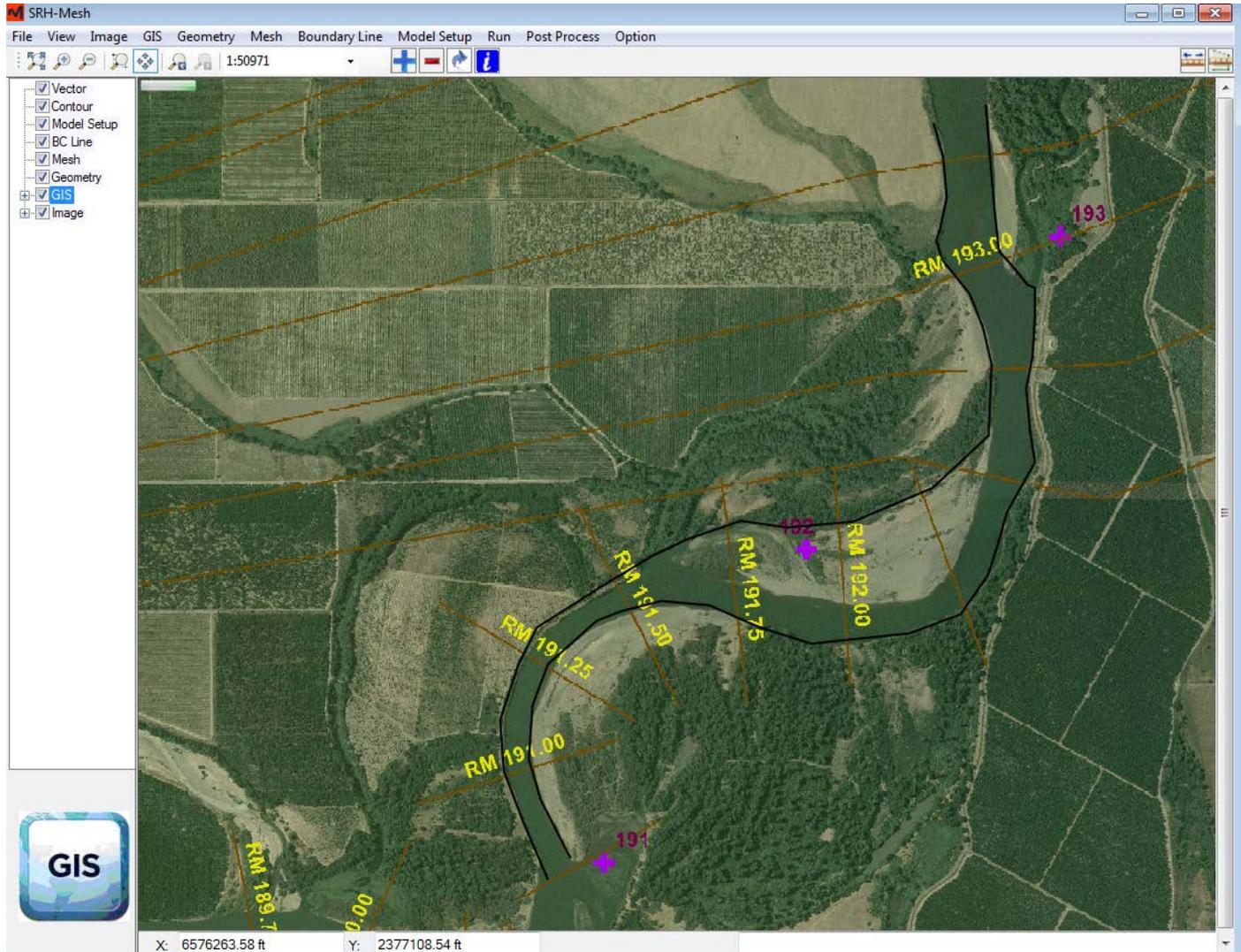
**Figure 5-4. Input GIS polylines.**

Step 7: Click the box immediately to the left of the GIS menu item in the table of contents to see the GIS file names. The GIS shapefile with name "boundary.shp" should be there. Check or uncheck the box immediately to the left of the GIS menu item in the table of contents GIS or immediately to the left of the GIS "boundary.shp" to show or hide all GIS shapefiles or each individual GIS shapefile.

Step 8: Click menu File\Save Project and save the project as "tutorial01.shrp".

## *5.2. Tutorial Two – Geometry Lines*

Tutorial Two shows how to create and edit geometry lines.

It starts from the saved project in Tutorial One. We will transfer the GIS shapefile into SRH-Mesh mesh and digitize new lines as upstream and downstream boundaries, edit vertexes, and redistribute vertexes in all the lines.

Step 1: Launch SRH-Mesh, open the file "tutorial01\tutorial01.srhp", and save this file as "tutorial02\tutorial02.srhp".

Step 2: Click on the menu item GIS\GIS Shapefile to Geometry to transform the GIS Shapefile into geometry features in SRH-Mesh. The line is shown as green lines in Figure 5-5. Uncheck the GIS Module in the table of contents.
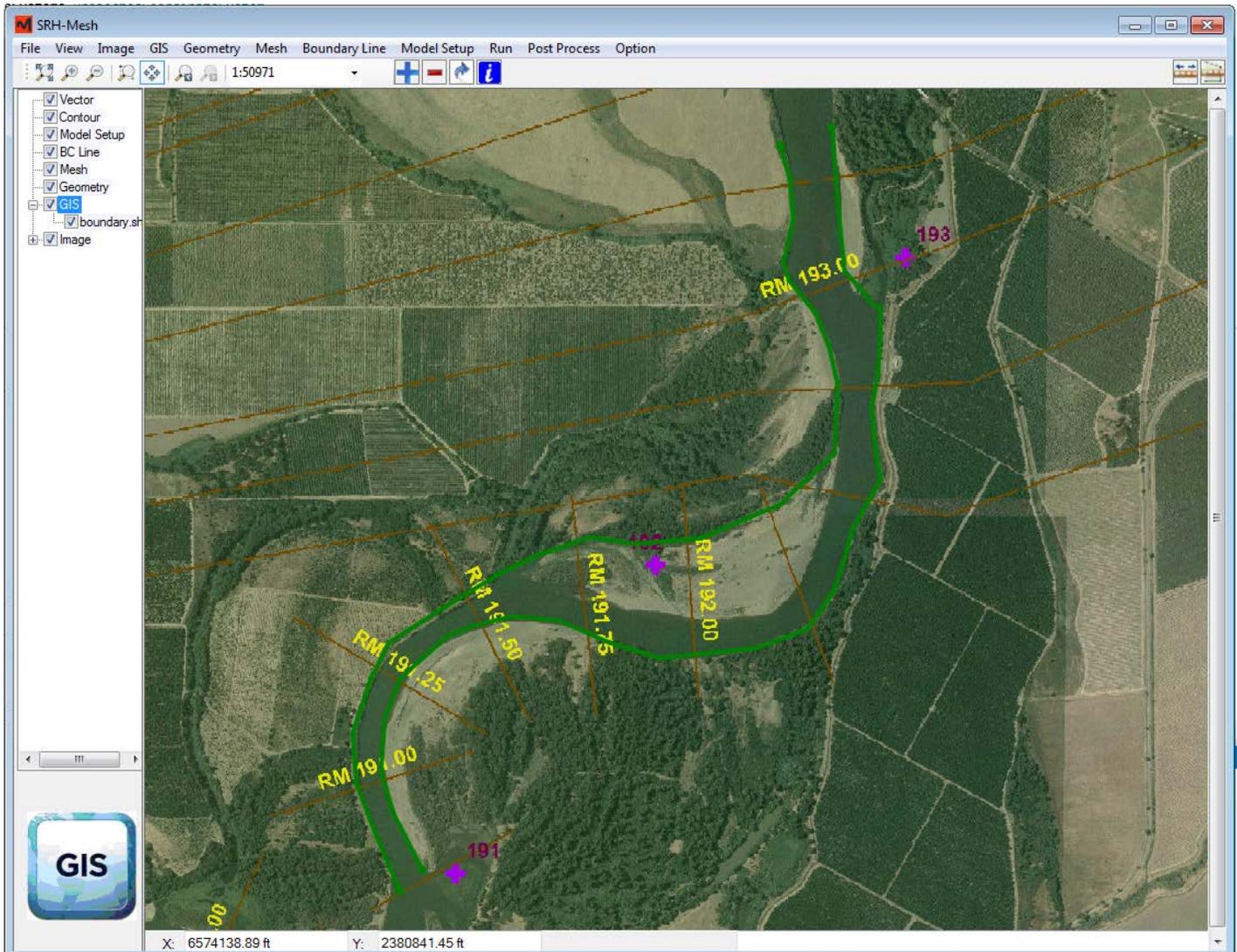


**Figure 5-5. SRH-Mesh mesh is shown as green lines.**

Step 3: Make sure that you are in the Geometry Module (the icon for the Geometry Module will appear in the lower left corner ). If not, click the Geometry Module in the table of contents.

Step 4: Click the Geometry Module menu and make sure that the Snap is On is checked and "snap distance = 20 pixels" (Section **Error! Reference source not found.**).

Step 5: Click the add button  in the Geometry Module toolbar to add a line. The Sacramento River flows from north to south, so the downstream end is at the bottom of your screen and the left bank should on your right side. Click as close as possible on the downstream of the left bank and move to the right side and double click on a point on right floodplain. The end point should be at least 20 pixels away from the downstream of the right bank and the new created line should be as perpendicular as possible to the two banks (Figure 5-6).
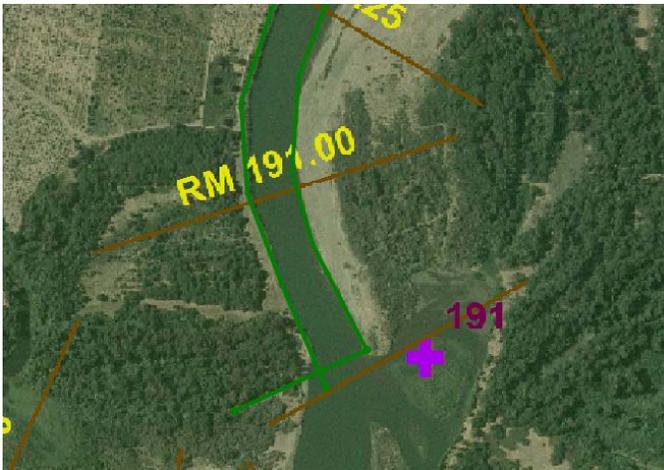


**Figure 5-6. New created line at downstream end**

Step 6. Follow Step 3 to create another line at the channel upstream end (Figure 5-7).

**Figure 5-7. New created line at upstream end.**

Step 7. Click the split button ⊤ in the Mesh Module toolbar to split the lines at intersections.

Step 8. Click the edit button ✎ in the Geometry Module toolbar, and the Edit toolbar will be shown on the right of the Geometry buttons. A copy of the Geometry lines will be displayed in blue. Select the lines outside of the polygon and click Delete in the Edit toolbar.

Step 9. Click the edit button ✎ in the Geometry Module toolbar to enter Edit mode again. Select the right bank, and perform the move, insert, and delete functions to make sure that the bank follows the GIS. Edit the left bank in the same way. Save and exit the Edit mode.

Step 10. Click the edit button ✎ in the Geometry Module toolbar to enter Edit mode again. Select the left and right banks and click Redistribute button in the Edit toolbar, a form will pop out. Set the Node Number as 50 nodes, click the yes button to close the Redistribute form.

Step 11. Follow Step 10 and redistribute the upstream and downstream lines with ten nodes. Save and exit the Edit mode.

Step 12: Select File\Save Project from the menu and save the project as "tutorial02.shrp".

## 5.3. Tutorial Three – Simple Mesh Generation

Tutorial three shows how to generate a series of simple 2D meshes. It starts from the saved project in tutorial two.

Step 1: Launch SRH-Mesh, open the file "tutorial02\tutorial02.srhp" and save as file "tutorial03\tutorial03.srhp". You can uncheck the boxes before GIS and GIS in the table of contents to hide the boxes.

Step 2: Make sure that you are in the Mesh Module ▯▷. If not, click Mesh in the table of contents.

Step 3: Click the select-outside-lines button ⬚ and hold the Ctrl key to select all four sides (lines) for the mesh. Click triangle-mesh button △, and you have your first triangle mesh (Figure 5-8).
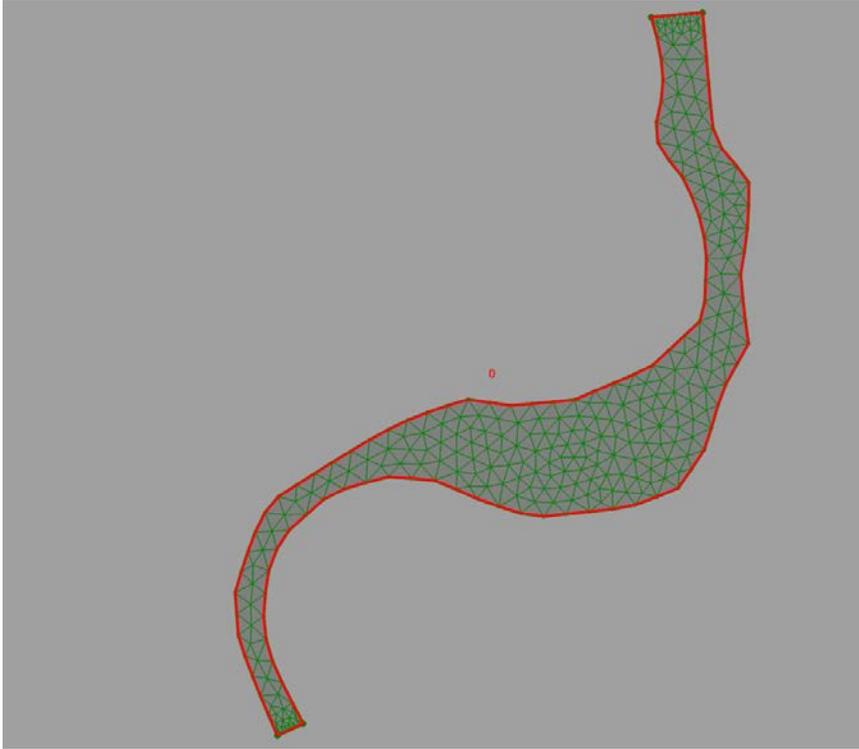


**Figure 5-8. SRH-Mesh triangle mesh.**

Step 4: Click the select mesh button ⬚ and then click a point inside of the mesh to select the mesh. Click the quadrilateral mesh button ⊞ , and you have your first quadrilateral mesh (Figure 5-9).
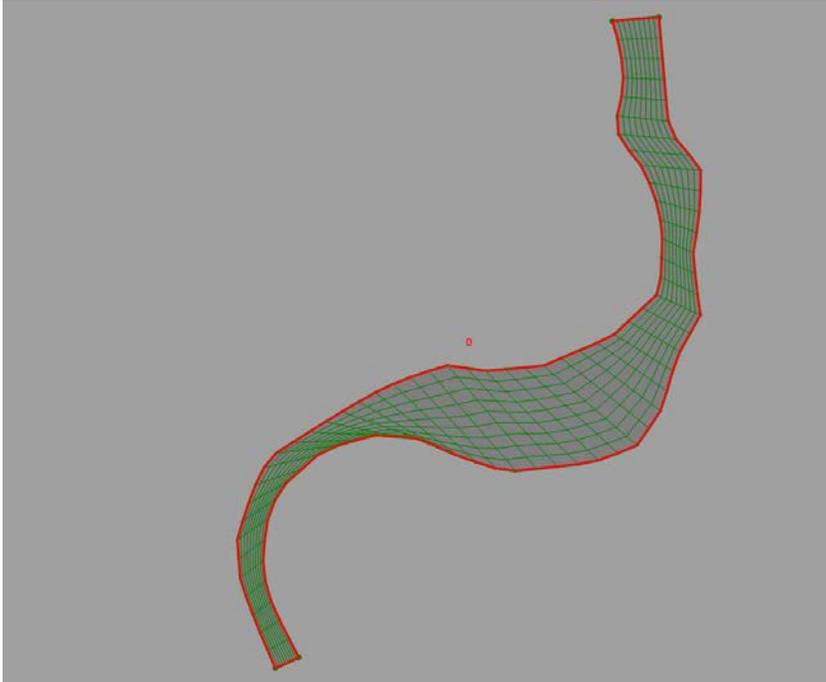
**Figure 5-9. SRH-Mesh quadrilateral mesh.**

## *5.4. Tutorial Four – Multi-Zone Mesh*

Tutorial four shows how to create a mesh with more than one zone and to assign the bed materials for roughness. It starts from the saved project in tutorial three. We will create a triangle mesh along the sandbar and a quadrilateral mesh along the mean channel. Inside the triangle mesh, we will create some denser mesh along a vegetated area.

Step 1: Launch SRH-Mesh, open the file "tutorial03\tutorial03.srhp", and save the file as "tutorial04\tutorial04.srhp".

Step 2: Go to Geometry Module, click the add button ✚ in the Geometry Module toolbar to add two lines: one along the channel near the point bar, one by the vegetation (Figure 5-10).
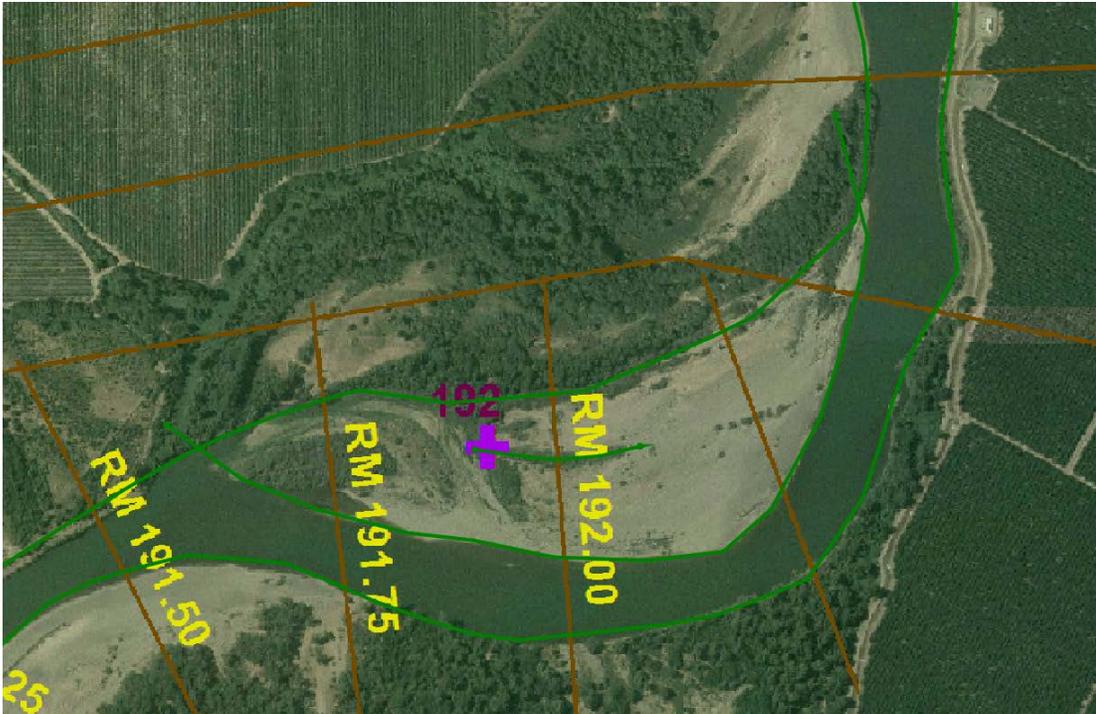
**Figure 5-10. Added lines near the sand bar.**

Step 3. Click the split button in the Geometry Module toolbar to split the lines at intersections. Go the Edit Mode to delete the line outside of the channel.

Step 4. Click the edit button in the Geometry Module toolbar, and the Merge/Split toolbar will be shown on the right of the Geometry Module toolbar. A copy of the mesh will be displayed in blue. Select the left bank and click Split at near the upstream and downstream of the point bar. Save and Exit Merge/Split toolbar.
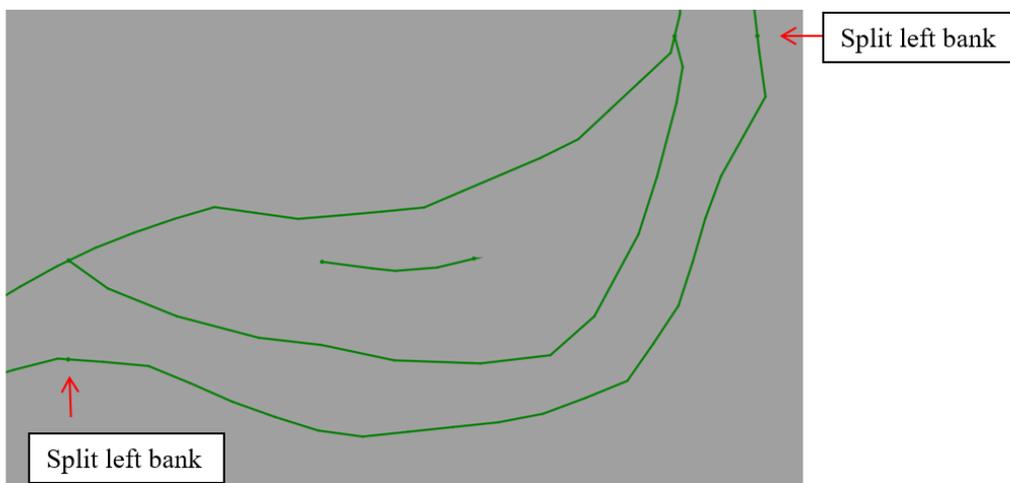


**Figure 5-11. Split left bank.**

Step 5. Click the edit line button ✎ in the Geometry Module toolbar to select lines and redistribute all 7 lines represent the banks and sand bar to 50 nodes, and the vegetation line into 20 nodes. Save and Exit.

Step 6: Go to mesh module. Click the select outside lines button 🔲 in the Mesh Module toolbar and hold the Ctrl key to select all two sides (lines) for around the point bar. Click the select inside lines button 🔲 and select the vegetation line. Click the triangle mesh button △, and you have a triangle mesh for the point bar.

Step 7: Click the select outside lines button 🔲 and hold the Ctrl key to select all bank lines and end lines for the channel. Click the quadrilateral mesh button ▱ , and you have your quadrilateral mesh for your channel (Figure 5-9).

Step 8: Right-click the Mesh Module in the table of contents (or use Menu Mesh\Set Mesh Prop erty). click reset change the bed zone display. Rename bed zones to Channel, Point Bar, and Vegetation. You can also change the display of bed zones according to your interest. Click the yes button to save the settings.

Step 9: Click the select mesh button 🔲 and then click a point inside the mesh for point bar to select the mesh. Click the set mesh roughness zone button ⬜. Change the bed material to 2, which is for a point bar. (Figure 5-12).
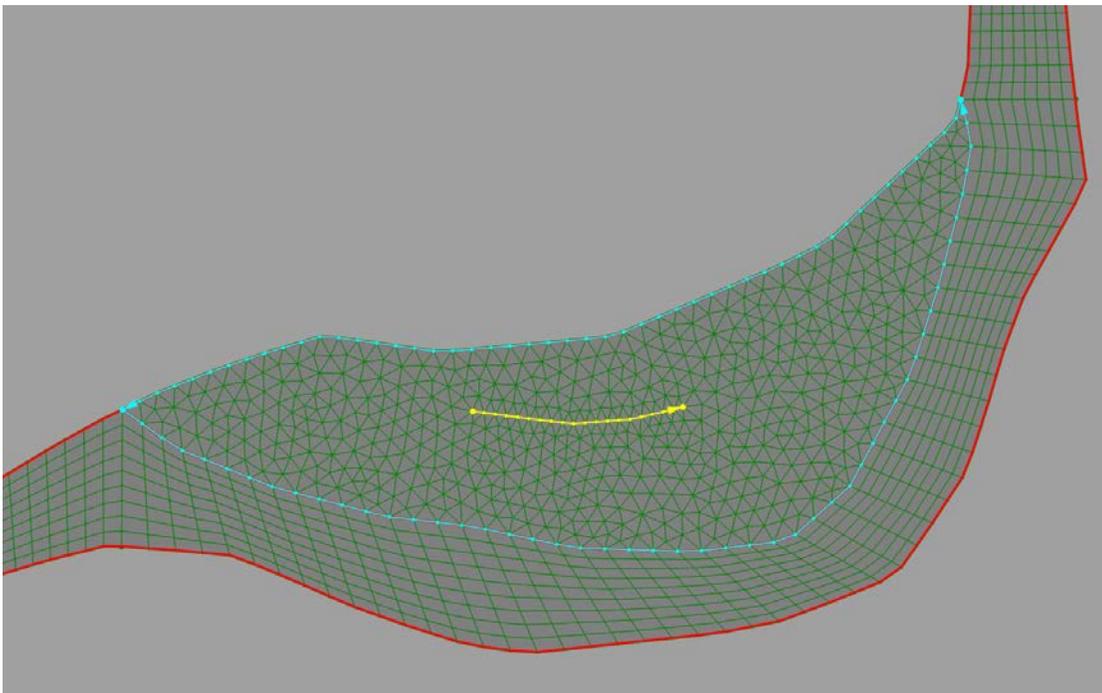


**Figure 5-12. Bed roughness for the point bar.**

Step 10: Click the select cell button 🔲 and then use your mouse to select the cells around the vegetation zone cells. Click to select one cell or click and drag to select all cells in the area. Press the Ctrl key and then make more selections. Click

the set cell roughness zone button ⬜, and change the bed material to 3, which is for vegetation zone. (Figure 5-13).
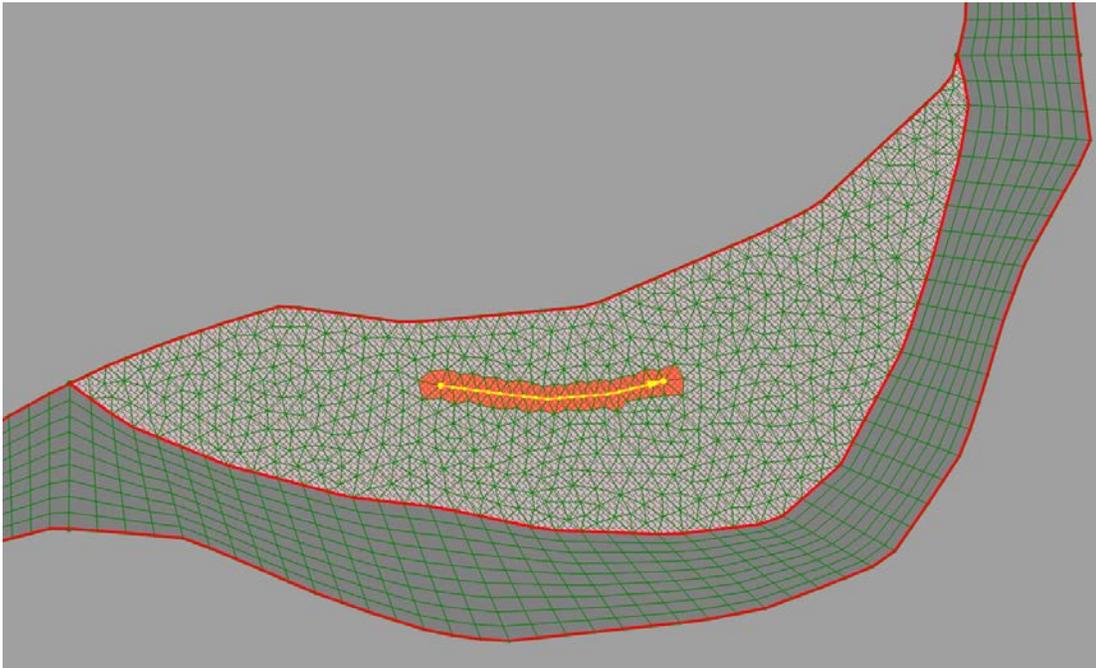


**Figure 5-13. Bed roughness for the vegetation zone.**

Step 11: Save the project.

## 5.5. Tutorial Five – Boundary Condition Line

Tutorial five shows how to create boundary condition lines. It starts from the saved project in tutorial four. We will create two boundary condition lines, one upstream and one downstream, as well as one monitoring line.

Step 1: Launch SRH-Mesh, open the file "tutorial04\tutorial04.srhp", and save the file as "tutorial04\tutorial04.srhp".

Step 2: Go to the Boundary Condition Line Module, click the add button ▷ in the Boundary Condition Line Module toolbar, click the vertex at top left corner of the mesh, and double-click the vertex of the top right corner of the mesh. Give the BC Line the name "Upstream" when prompted (Figure 5-14).
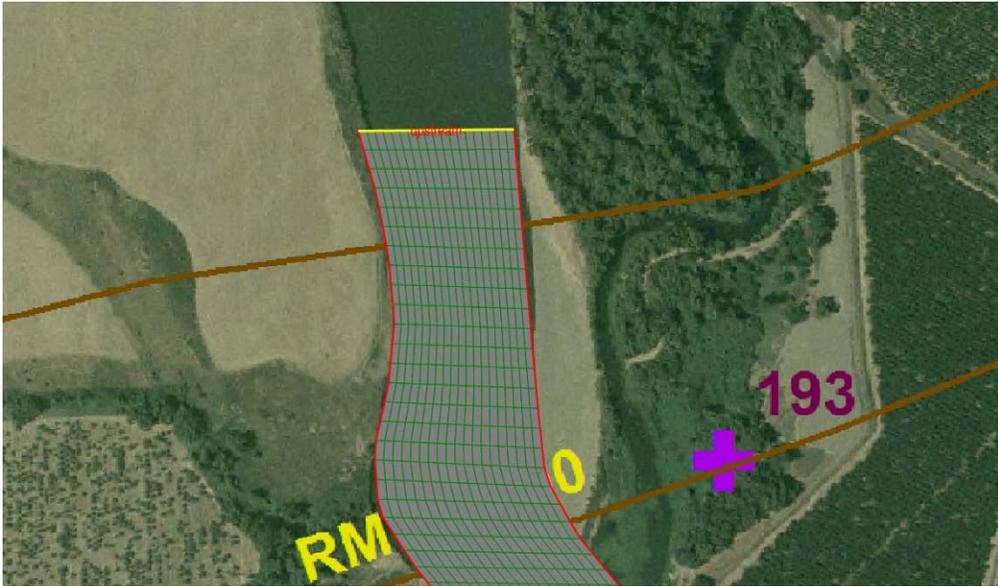
**Figure 5-14. Upstream boundary condition line.**

Step 3. Move the map to the downstream of the channel, click the add button ⬗ in the Boundary Condition Line Module  toolbar, click the vertex at bottom left corner of the mesh, and double-click the vertex of the bottom right corner of the mesh. Give the BC Line the name "Downstream" when prompted (Figure 5-15).



**Figure 5-15. Downstream boundary condition line.**

Step 4. Move the map to the downstream end of the point bar, click the add monitor line button ⬗ in the Boundary Condition Line Module toolbar, click the vertex at one side of the channel and double-click the vertex of the other side of the channel. Give the Monitoring Line the name "Monitor" when prompted (Figure 5-16).
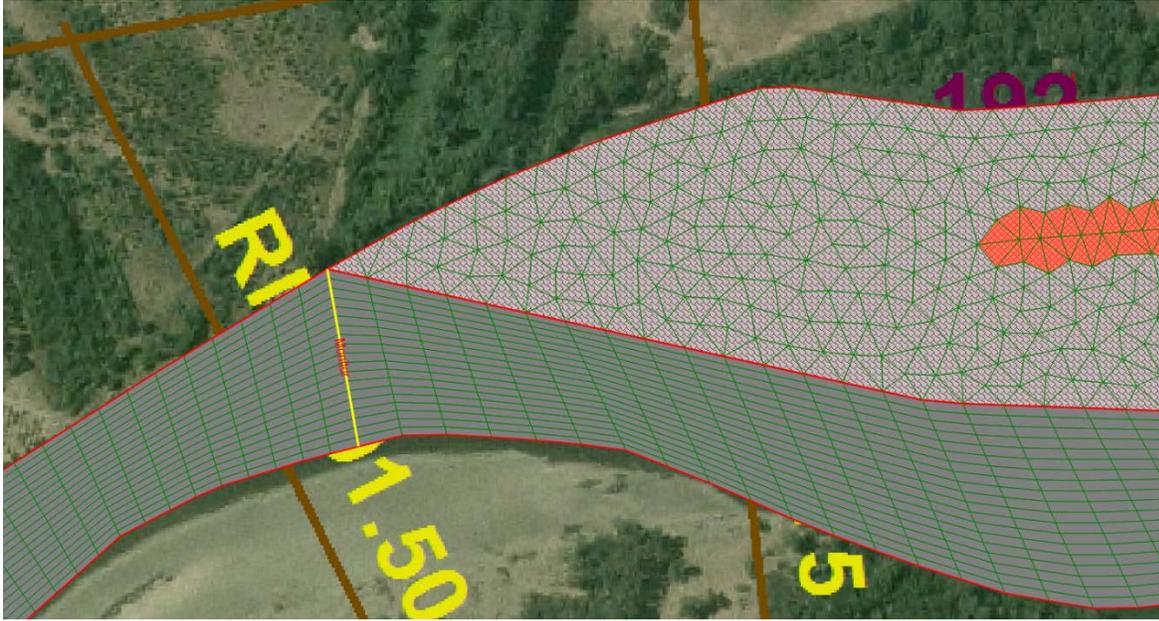
**Figure 5-16. Monitoring line.**

Step 5: Click the save button 🖫 to save mesh and boundary condition lines in ".srhm" format.

Step 6: Save the project.

# Appendix A. Example SRH-Mesh Mesh File

## A.1. Header

A SRH-Mesh mesh file (.srhm) starts with two lines of a header. For exam    ple:

```
SRH-Mesh
NAME "2D unstructure mesh"
```

## A.2. Vertex

It is then followed by the vertex data, starting with number of vertexes and x, y, z coordinates for each vertex. For example:

```
VERTEX 60
6.5722720742E+006 2.3728513889E+006 0.0000000000E+000
6.5718524273E+006 2.3738811341E+006 0.0000000000E+000
6.5718153926E+006 2.3749636725E+006 0.0000000000E+000
…
```

In this example, a total of 60 vertexes are defined. The first vertex (ID=0) is located at x=6.5718153926E+006, y=2.3749636725E+006, and z=0.0000000000E+000.

## A.3. Cell

Cell data starts with the first line that defines the number of cells, then one data line for a cell that contains the number of vertexes in the cell, vertex indexes of the cell, and then roughness index. Followed by the optional data for bed gradation zone index and initial boundary condition zone. For example:

```
CELL 44 4 False False
 4 0 13 14 1 0
 4 1 14 15 2 0
 4 2 15 16 3 0
 4 3 16 17 4 0
 4 4 17 18 5 0
 …
```

In this example,

> CELL 44 4 False False:
>> CELL: cell data set
>> 44: a total of 44 cells
>> 4: maximum number of vertexes in a cell.
>> False: No bed gradation zone is defined.
>> False: No initial boundary condition is defined
> 4 0 13 14 1 0
>> 4: first number is the total number of vertexes in the cell.
>> 0 13 14 1: indexes of vertex ID in the cell with the total number
>>> defined by the first number (4).
>> 0: roughness zone of the cell.
> 4 1 14 15 2 0

4: first number is the total number of vertexes in the cell.
1 14 15 2: indexes of vertex ID in the cell with the total number
defined by the first number (4).
0: roughness zone of the cell.

## A.5. BC Line

BC Line data starts with the first line that defines the number of BC Line lines. For each BC Line line, it starts with a line that defines the number of vertexes in the line, followed by vertex indexes for the line. For example:

```
LINE 2
LN 0 4 "upstream"
 12 25 38 59
LN 1 5 "downstream"
 0 13 26 39 47
```

In this example,

LINE 2: Two BC Lines are defined.
LN 0 4 "upstream": the first line (ID=0) has 4 vertexes with a name "upstream".
The line vertex indexes are listed as: 12 25 38 59
LN 1 5 "downstream": the second line (ID=1) has 5 vertexes with a name "downstream".
The line vertex indexes are listed as: 0 13 26 39 47


## A.7. END

END: End record.


## A.8. Example SRH-Mesh Mesh Data

```
SRH-Mesh
NAME "2D unstructure mesh"
VERTEX 60
6.5722720742E+006 2.3728513889E+006 0.0000000000E+000
6.5718524273E+006 2.3738811341E+006 0.0000000000E+000
6.5718153926E+006 2.3749636725E+006 0.0000000000E+000
6.5723211193E+006 2.3759219856E+006 0.0000000000E+000
6.5732639953E+006 2.3765119967E+006 0.0000000000E+000
6.5742628380E+006 2.3769934185E+006 0.0000000000E+000
6.5753466621E+006 2.3770603746E+006 0.0000000000E+000
6.5764218148E+006 2.3772878135E+006 0.0000000000E+000
6.5773521582E+006 2.3778666124E+006 0.0000000000E+000
6.5777010730E+006 2.3788388872E+006 0.0000000000E+000
6.5774232331E+006 2.3798993204E+006 0.0000000000E+000
6.5771238758E+006 2.3808813903E+006 0.0000000000E+000
6.5770019773E+006 2.3819711381E+006 0.0000000000E+000
6.5723512560E+006 2.3728934073E+006 0.0000000000E+000
6.5719377671E+006 2.3738845619E+006 0.0000000000E+000
6.5719023621E+006 2.3749406125E+006 0.0000000000E+000
```

```
6.5724157415E+006 2.3758717256E+006 0.0000000000E+000
6.5733666873E+006 2.3764150909E+006 0.0000000000E+000
6.5743829470E+006 2.3767196363E+006 0.0000000000E+000
6.5754737160E+006 2.3767160597E+006 0.0000000000E+000
6.5765714023E+006 2.3769369272E+006 0.0000000000E+000
6.5774794755E+006 2.3775061319E+006 0.0000000000E+000
6.5778606466E+006 2.3785546002E+006 0.0000000000E+000
6.5776820647E+006 2.3797008276E+006 0.0000000000E+000
6.5773576126E+006 2.3807785763E+006 0.0000000000E+000
6.5772198586E+006 2.3819734360E+006 0.0000000000E+000
6.5724304377E+006 2.3729354257E+006 0.0000000000E+000
6.5720231068E+006 2.3738879896E+006 0.0000000000E+000
6.5719893315E+006 2.3749175525E+006 0.0000000000E+000
6.5725103636E+006 2.3758214656E+006 0.0000000000E+000
6.5734693792E+006 2.3763181850E+006 0.0000000000E+000
6.5745030560E+006 2.3764458541E+006 0.0000000000E+000
6.5756007699E+006 2.3763717447E+006 0.0000000000E+000
6.5767209897E+006 2.3765860410E+006 0.0000000000E+000
6.5776199898E+006 2.3771526544E+006 0.0000000000E+000
6.5780301179E+006 2.3782755656E+006 0.0000000000E+000
6.5779474948E+006 2.3795058363E+006 0.0000000000E+000
6.5775946486E+006 2.3806775131E+006 0.0000000000E+000
6.5774377398E+006 2.3819757339E+006 0.0000000000E+000
6.5725096194E+006 2.3729774441E+006 0.0000000000E+000
6.5721084466E+006 2.3738914175E+006 0.0000000000E+000
6.5720763010E+006 2.3748944925E+006 0.0000000000E+000
6.5726049858E+006 2.3757712057E+006 0.0000000000E+000
6.5735720712E+006 2.3762212792E+006 0.0000000000E+000
6.5746231650E+006 2.3761720720E+006 0.0000000000E+000
6.5757278238E+006 2.3760274299E+006 0.0000000000E+000
6.5768705772E+006 2.3762351548E+006 0.0000000000E+000
6.5725888011E+006 2.3730194624E+006 0.0000000000E+000
6.5721937863E+006 2.3738948452E+006 0.0000000000E+000
6.5721632704E+006 2.3748714325E+006 0.0000000000E+000
6.5726996080E+006 2.3757209457E+006 0.0000000000E+000
6.5736747632E+006 2.3761243732E+006 0.0000000000E+000
6.5747432740E+006 2.3758982897E+006 0.0000000000E+000
6.5758548777E+006 2.3756831149E+006 0.0000000000E+000
6.5770201647E+006 2.3758842685E+006 0.0000000000E+000
6.5777605041E+006 2.3767991769E+006 0.0000000000E+000
6.5781995892E+006 2.3779965310E+006 0.0000000000E+000
6.5782129249E+006 2.3793108450E+006 0.0000000000E+000
6.5778316846E+006 2.3805764498E+006 0.0000000000E+000
6.5776556211E+006 2.3819780317E+006 0.0000000000E+000
CELL 44 4 False False
 4 0 13 14 1 0
 4 1 14 15 2 0
 4 2 15 16 3 0
 4 3 16 17 4 0
 4 4 17 18 5 0
 4 5 18 19 6 0
 4 6 19 20 7 0
 4 7 20 21 8 0
```

```
4 8 21 22 9 0
4 9 22 23 10 0
4 10 23 24 11 0
4 11 24 25 12 0
4 13 26 27 14 0
4 14 27 28 15 0
4 15 28 29 16 0
4 16 29 30 17 0
4 17 30 31 18 0
4 18 31 32 19 0
4 19 32 33 20 0
4 20 33 34 21 0
4 21 34 35 22 0
4 22 35 36 23 0
4 23 36 37 24 0
4 24 37 38 25 0
4 26 39 40 27 0
4 27 40 41 28 0
4 28 41 42 29 0
4 29 42 43 30 0
4 30 43 44 31 0
4 31 44 45 32 0
4 32 45 46 33 0
3 33 46 34 0
4 39 47 48 40 0
4 40 48 49 41 0
4 41 49 50 42 0
4 42 50 51 43 0
4 43 51 52 44 0
4 44 52 53 45 0
4 45 53 54 46 0
4 46 54 55 34 0
4 34 55 56 35 0
4 35 56 57 36 0
4 36 57 58 37 0
4 37 58 59 38 0
LINE 2
LN 0 4 "upstream"
 12 25 38 59
LN 1 5 "downstream"
 0 13 26 39 47
END
```