SRH and HEC RAS - 2D Hydraulic Modeling Workshop
## Course Agenda

### Day 1: Wednesday, March 1st

<table>
<thead>
<tr>
<th>Lesson: SRH-2D</th>
<th>Tentative Duration (minutes)</th>
<th>Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction and Course Overview</td>
<td>60</td>
<td>9:00 AM – 10:00 AM</td>
</tr>
<tr>
<td>Modeling Exercise #1 - Background Data – Part A</td>
<td>60</td>
<td>10:00 AM – 11:00 AM</td>
</tr>
<tr>
<td>Modeling Exercise #2 - Background Data – Part B</td>
<td>45</td>
<td>11:00 AM – 11:40 AM</td>
</tr>
<tr>
<td>Lunch</td>
<td>90</td>
<td>12:00 PM – 1:30 PM</td>
</tr>
<tr>
<td>Modeling Exercise #3 - Parameters</td>
<td>90</td>
<td>1:30 PM – 3:00 PM</td>
</tr>
<tr>
<td>Break</td>
<td>15</td>
<td>3:00 PM – 3:15 PM</td>
</tr>
<tr>
<td>Modeling Exercise #4 - Geometry</td>
<td>45</td>
<td>3:15 PM – 4:00 PM</td>
</tr>
<tr>
<td>Modeling Exercise #4 - Simulation</td>
<td>30</td>
<td>4:00 PM – 4:30 PM</td>
</tr>
</tbody>
</table>

### Day 2: Thursday, March 2nd

<table>
<thead>
<tr>
<th>Lesson: HEC RAS-2D</th>
<th>Tentative Duration (minutes)</th>
<th>Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modeling Exercise #1 - ArcMap</td>
<td>90</td>
<td>9:00 AM – 10:30 AM</td>
</tr>
<tr>
<td>Break</td>
<td>15</td>
<td>10:30 AM – 10:45 AM</td>
</tr>
<tr>
<td>Modeling Exercise #2 - Background Data</td>
<td>15</td>
<td>10:45 AM – 11:00 AM</td>
</tr>
<tr>
<td>Modeling Exercise #3 - Development of a 2D Model</td>
<td>60</td>
<td>11:00 AM – 12:00 PM</td>
</tr>
<tr>
<td>Lunch</td>
<td>90</td>
<td>12:00 PM – 1:30 PM</td>
</tr>
<tr>
<td>Modeling Exercise #4 - Unsteady Flow Model</td>
<td>30</td>
<td>1:30 PM – 2:00 PM</td>
</tr>
<tr>
<td>SRH Modeling Exercise Extra – Quartz Creek</td>
<td>60</td>
<td>2:00 PM – 3:00 PM</td>
</tr>
<tr>
<td>Break</td>
<td>15</td>
<td>3:00 PM – 3:15 PM</td>
</tr>
<tr>
<td>Open Discussion Among Participants (Model Comparison, User Friendly, Data Requirements ...)</td>
<td>75</td>
<td>3:15 PM – 4:30 PM</td>
</tr>
</tbody>
</table>
Day 1

2D Hydraulic Modeling of Rivers using SRH-2D
### SRH-2D Table of Contents

**Modeling Exercise #1 – Background Data – Part A**

- Gathering Data ....................................................................................................................................... 1-1
  1. Introduction ...................................................................................................................................... 1-1
  2. Projection ......................................................................................................................................... 1-2
  3. Digital Elevation Data ..................................................................................................................... 1-2
  4. Display Options ............................................................................................................................. 1-3
  5. Background Image .......................................................................................................................... 1-5
  6. Hydraulic Structures - Bridge Piers ............................................................................................. 1-10
  7. Conclusion ..................................................................................................................................... 1-11

**Modeling Exercise #2 – Background Data – Part B**

- Evaluating and Editing Data ................................................................................................................. 1-12
  1. Introduction ..................................................................................................................................... 1-12
  2. Cleaning Up the Triangulations ..................................................................................................... 1-12
  3. Breaklines ......................................................................................................................................... 1-15
  4. Conclusion ..................................................................................................................................... 1-18

**Modeling Exercise #3 – Parameters**

- Building Feature Objects .................................................................................................................... 1-19
  1. Introduction ..................................................................................................................................... 1-19
  2. Initializing the SMS Workspace ...................................................................................................... 1-19
  3. Using Feature Objects .................................................................................................................... 1-19
  4. Mesh Generator Coverage ............................................................................................................. 1-19
  5. Boundary Conditions Coverage .................................................................................................... 1-24
  6. Material Coverage .......................................................................................................................... 1-28
  7. Monitor Points Coverage ................................................................................................................ 1-31
8. Conclusion ................................................................................................................................ 1-32

Modeling Exercise #4 – Geometry ........................................................................................................... 1-33

Generating a Mesh................................................................................................................................... 1-33

1. Introduction ...................................................................................................................................... 1-33

2. Subdividing the Domain ................................................................................................................... 1-33

3. Model Resolution ............................................................................................................................. 1-33

4. Meshing Parameters ......................................................................................................................... 1-34

5. Assigning Materials .......................................................................................................................... 1-40

6. Generating the Mesh ........................................................................................................................ 1-40

7. Evaluating the Mesh ........................................................................................................................ 1-42

8. Conclusion ...................................................................................................................................... 1-43

Modeling Exercise #5 – Model Simulation .................................................................................................. 1-44

SRH-2D Simulation Setup and Model Run .............................................................................................. 1-44

1. Introduction ...................................................................................................................................... 1-44

2. Creating a Simulation ....................................................................................................................... 1-44

3. Linking Components ......................................................................................................................... 1-44

4. Define Model Control ....................................................................................................................... 1-45

5. Running the Model .......................................................................................................................... 1-46

6. Post-Processing ................................................................................................................................ 1-48

7. Map Export ...................................................................................................................................... 1-49

8. Create Profile Plots ............................................................................................................................ 1-50

9. Create Animation ............................................................................................................................. 1-50

10. Conclusion ...................................................................................................................................... 1-51
Modeling Exercise #1 – Background Data – Part A

Gathering Data

1. Introduction

This workshop is based on real data collected by the State of Alaska Department of Transportation and Public Facilities (AKDOT&PF) for a bridge replacement over the Lakina River on the McCarthy Road (Figure 1).

![Figure 1: Project Location Map for the Lakina River Bridge Replacement. Image from Lakina River Bridge Right of Way Plans.](image)

This exercise illustrates a typical set of steps to import background data used when beginning a hydrodynamic modeling project in SMS including surveyed xyz data and base maps.

The data files for this workshop are located in the “Workshop” folder.

Launch SMS from the desktop icon, start menu or from file browser. SMS will open with a display window.
2. **Projection**

In an SMS project, the coordinate system used to represent the geographic data is referred to as a projection. Each data object, whether it is an image, a survey, a CAD file, or any other geographic data is referenced to a projection.

For this project, the projection can be found using the DOT plans for the Lakina River Bridge Right of Way (ROW), which is located in the “Workshop\DOT Plans” folder. Using the Lakina River plans, find the following information:

- Projection: ________________________________________________________
- Zone: ____________________________________________________________
- Datum: ___________________________________________________________
- Planar Unit: _______________________________________________________

Set the Display Projection for the Lakina River using the following steps:

- Select *Display | Projection | Global Projection | Set Projection*...
- Fill in the recently acquired information.

*Note: The Vertical and Horizontal units need to be the same: Feet (U.S. Survey).

- Click OK to exit the dialog.

The projection to be used for this SMS project has now been defined.

Save the project:

- Select *File | Save New Project*... enter a name of “Lakina River” and click save.

*Note: SMS does not have an “undo” option, so it is important to save frequently.

3. **Digital Elevation Data**

The first source of geometry data for this project is a scatter set. A scatter set is a set of points at any x,y location. Each point has one data value. For this exercise, this data value represents an elevation. The scatter set was collected by ground based survey crews. This process is time consuming and often results in pore resolution datasets.

To load the elevation data:

- Select *File | Open* or use the *Open Shortcut* tool 📝. The open dialog will appear. Select the file LakinaRiver.dwg in the “Workshop\DOT Scatter Data\AutoCAD” folder.
This data file is from AutoCAD and includes a lot of data layers. Quickly play around with the dataset to see if you can display just the elevation scatter data. Don’t spend too much time on this, as the point of bringing in the AutoCAD file is to demonstrate that some file types are easier to work with than others.

Let’s delete the AutoCAD file and bring in an Extensible Markup Language (XML) instead. XML files are plain text files and are similar to HTML.

- To delete the file, R-Click LakinaRiver.dwg in the Project Explorer tree and select Delete.
- To open the XML file, Select Open then select the LakinaRiver in the “Workshop\DOT Scatter Data\XML” folder.
  - The layer can be renamed by R-Clicking on the scatter data “EG – 63905” in Project Explorer and selecting Rename. Rename to “Lakina Scatter”.

**4. Display Options**

We are now going to play around with different Display Options:

- To zoom in to the layer:
  - R-Click scatter data in project explorer| Zoom to Scatter.
- To view scatter points:
  - Click Display in the Menu Bar | Display Options. This will cause the Display Options dialog to appear. Next time we will use the Display Options Shortcut tool.
    - Make sure Scatter is highlighted.
    - A message might appear saying “Too many contour values to show. Contours options will be change to 1000 contours.”
    - Click OK, this just auto adjust the contour interval.
  - Select ✔ Points and un-selected □ Contours. Then Click OK. You should now be able to see all the data points.
- To turn the contours back on and change the contour method to include contour lines:
  - Select the Display Options Shortcut tool, make sure Scatter is highlighted, un-select □ points and select ✔ Contours.
Then choose the Contours Tab. In the Contour method drop down, choose Color Fill and Linear. Then Click OK.

Contour line should now be visible, but there might be too many of them.

To change the contour interval and color ramp:

Select the Display Options Shortcut tool, make sure Scatter is highlighted and under the Contours Tab the Contour Interval can be changed. Try selecting Specified Interval from the drop down menu, then enter in 1.5 for the contour interval.

There are also lots of color scheme possibilities. Play around with Color Ramp and subsequent color options. Click OK when finished.

The “Rotate” tool found in the Dynamic Tool bar is a great way to see if the terrain makes sense. Once the rotate tool is selected, click on the screen and move your mouse around to get the contour map to illustrate the topography (Figure 2).

To make the elevation change more exaggerated, the Z magnification can be changed. To do so, select the Display Options Shortcut tool | General | and enter 3 or 4 into the Z magnification space. Check OK to exit.

To get back to the original overhead view, select the Plan View tool.

SAVE Project.

Figure 2: Image of Lakina River contours rotated to illustrate topography.
5. Background Image

A good way to help visualize the model is to import a digital image of the site. This image may be an aerial photo or a topographic chart. In this exercise, an aerial photo will be used.

*SMS supports several different image formats. Some of the more common ones include JEIG, TIF, PNG, BMP, SID, and ECW. Images may be available from online maps services, government sites or obtained locally.*

5.1. Loading an Image

To load an image of the site:

- Select Open then select the Lakina_13 image in the “Workshop\DOT Images\Lakina River” folder.

A Register Image dialog window should pop up (Figure 3) because the image file is not georeferenced. Before an image can be displayed, the image must be "registered" or georeferenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window.

![Figure 3: Register Image Dialog.](image-url)
5.2. Registering Images

Register Image Dialog:

An image is registered using the Register Image dialog. The main feature of the Register Image dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- 2 point or 3 point registration – Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- Import World File – Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.

Register Image Dialog Tools: The following tools can be used to help position the registration points (Table 1):


<table>
<thead>
<tr>
<th>Tool</th>
<th>Tool Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>🔍</td>
<td>Select Point Tool</td>
<td>The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.</td>
</tr>
<tr>
<td>🔍</td>
<td>Zoom Tool</td>
<td>In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom tool is used to zoom in a portion of the image.</td>
</tr>
<tr>
<td>🔄</td>
<td>Pan Tool</td>
<td>After zooming in on a portion of the image, the Pan tool is used to pan the image vertically or horizontally.</td>
</tr>
<tr>
<td>✚</td>
<td>Frame Macro</td>
<td>The Frame macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.</td>
</tr>
</tbody>
</table>

Before we begin the image registration process we need some real world coordinates (X,Y), so let’s Cancel out of the Register Image dialog and get some. Cancel again to fully escape.

Acquiring real world coordinates (X,Y) from our dataset is easiest if the scatter points are turned on.
• Select the Display Options Shortcut tool, make sure Scatter is highlighted, then select Points. Check OK to exit.

The next step involves selecting 3 scatter points, writing down their associated coordinates, and making approximate image location notes for where the points are located. Successfully registering the image will require many iterations of this process.

• Click on the scatter data Lakina River to make sure it is highlighted in the Project Explorer.

• Using the Select Scatter Points tool select three points of interest. Record the X and Y coordinate information below. The coordinates for the selected points appears at the top of the screen. The coordinates are also provided at the bottom of the screen, but those values change with the movement of the mouse while the top of the screen values stay fixed for the selected point.

  o Point 1:
    ▪ X:
    ▪ Y:
    ▪ Description:

  o Point 2:
    ▪ X:
    ▪ Y:
    ▪ Description:

  o Point 3:
    ▪ X:
    ▪ Y:
    ▪ Description:

Make detailed location notes for the three points, i.e. Point 3 is located at the very SE corner of the bridge, or point 1 is located west of the bridge at intersection of main road and spur road that heads north (Figure 4). It will help to have the image open using an image viewer while you complete this step. Spread the points out for better scaling of the image.
Figure 4: A) Two selected points of interest are represented by yellow dots (\(\bullet\)). B) Corresponding to the yellow dots in image A, location notes need to be made so the image can later be registered.

Now that we have some reference points, we can reattempt to register the image:

- If the “Lakina_13.tif” image is under the GIS Data in the Project Explorer, then R-Click the image and select Register Image...

- If the image is not present, then it needs to be reopened:
  - Select Open then select the Lakina_13 image in the “Workshop\DOT Images\Lakina River” folder.

- In the Register Image dialog use the Select pointer tool to move the three points (shown by “+” symbols) to the location of your three selected scatter points. Use the Zoom tool to zoom in for a better location placement. The Pixel coordinates (X and Y) should change as you move the points around.

- Once the three points (+ symbols) are in position, fill in the real world coordinates (X,Y) into the World Coordinate locations.

- When finished, click OK.

Voila! The image should be somewhere near your scatter points. I assume it’s not perfect the first time. Registering images can be a very time consuming and tedious process.

By making the contours transparent, the alignment can be better seem:

- Select the Display Options Shortcut tool and navigate to the Contours tabs. Make the Transparency 50% and click OK.

- Zoom to Scatter.
Instead of wasting time making the alignment perfect, we will move on to import a prepared World File.

5.3. **World File**

By itself, an image does not correspond explicitly to any point on a map or location in a project. The process of aligning the image with a projection is known as georeferencing.

A common and simple way of georeferencing uses an extra file known as a “world file”. This file may have an extension of “.wld” indicating it is a world file.

In the *Register Image* dialog, there is an *Import World* File button that allows you to bring in a world file associated with a previously registered image. The file contains registration data that can be used to register the image.

![Figure 5: Successfully geo-referenced image.](image)
- R-Click on the “Lakina_13.tif” image under the GIS Data in the Project Explorer and select Register Image...
  - Select Import World File... down at the bottom.
  - Navigate to Workshop\DOT Images\Lakina River\Lakina_13 World and select Lakina 13.wld and click Open.
  - Finally click OK and the image should be aligned with our dataset (Figure 5).
  - Zoom to Scatter.
  - SAVE Project.

6. Hydraulic Structures - Bridge Piers

The Lakina River Bridge has spill-through abutments that are represented in the topo data, but there are six piers that still need to be represented. The piers are 3-ft diameter piles. The coordinate information is provided in an excel spreadsheet located in the “Bridge Data” folder.

- Open the “Substructures” file in excel to view the data. There are three tabs: Both, Quartz, and Lakina. Briefly familiarize yourself with spreadsheet.
  - Save the Lakina tab as a .txt file.
    - While the Lakina tab is open select File | Save As:
      - Change the Save as type to “Text (Tab delimited)” using the drop down menu.
    - Click save.

To import the Bridge data into SMS follow the subsequent steps:

- Select Open and navigate to your .txt file and Open it.
- Select Use Import Wizard | OK.
- The data is Tab delimited, so make sure Tab is selected.
- Also make sure Start import at row is “1” and Heading row is selected, then click Next >.
- Make sure the Type is correct... X for X and Y for Y, then click finish.
• Using the Zoom tool or the Zoom to Scatter command for the Substructures layer, zoom in to see if the piles look to be in the correct locations (Figure 6).

• You can turn off the contours by selecting the Display Options Shortcut tool, make sure Scatter is highlighted and un-selecting Contours. Check OK to exit.

• SAVE Project.

Figure 6: Lakina River Bridge Piers represented by red dots.

The river channel and gravels are constantly in flux and the background image outdates the survey data. For a more current view, the new bridge can be seen in images located in this folder: Workshop\DOT Images\Lakina River\New Bridge at Lakina River.

7. Conclusion

This exercise illustrated how to initialize a project, set up projections, import in CAD data and elevation data, import in and geo-reference images, import in world and text files.
1. Introduction

This exercise illustrates a typical set of steps to modify triangulation, manually and with breaklines.

Launch SMS from the desktop icon, start menu or from file browser. To ensure consistency, read in a set of completed files from the end of exercise #1.

- Open SMS and select Open.
- Navigate to “SMS Modeling Exercises\Exercise 1 – Background Data A” directory
- Select the file named “Lakina River.sms.”, then select Open.

If it asks “Do you want to delete existing data?,” Click Yes.

2. Cleaning Up the Triangulations

SMS triangulates the scatter points creating a triangulated irregular network (TIN) (Figure 7). This triangulation creates a surface, or finite element mesh using all the points. For many cross section datasets this results in large areas inside of river meander bends included in the TIN which have no data points and therefore do not represent the elevation. To clean this up the triangles in these areas must be deleted. Cleaning up the triangulation can be done manually, but it also can be done by selection a maximum edge length.

![Figure 7: Triangulated Irregular Network (TIN).]
To see if we need to clean up the Lakina River TIN, we first need to turn on the triangles.

- Select the Display Options Shortcut tool, make sure Scatter is highlighted, then select All Off at the bottom and select Triangles. Check OK to exit.

- Click on the scatter data Lakina River to make sure it is highlighted in the Project Explorer. And un-select the Substructures scatter data.

- Zoom to Scatter.

- The background image can be left on or turned off by un-selecting Lakina_13.tif.

Our area of interest for the Lakina River channel is relatively straight, so there are not excessive areas in the floodplain without data. There are large triangle in the hills they are not relevant as the flood water will not be modeled that high.

There is, however, a major problem with the TIN, but it is hard to see in this view.

Using contours (Color Fill and Linear), reversing the color ramp, the rotate tool, and a Z-magnification of 4, Figure 8 was made to illustrate the problem. As a result of the triangulation process, the downstream boundary has an artificial lip at the end, which acts as a wall. This bogus boundary will cause the prevention of modeled flow.

![Figure 8: Lakina River contours map illustrating artificial lip, which is defined within the red circle.](image_url)
To fix this problem, we need to manually clean up the downstream triangulations. A zoomed in view of the triangles can be seen in Figure 9. We need to remove the triangles which form a wall in the middle of the channel.

- SAVE Project . Remember SMS does not have an “undo” option, so be careful during this process. If you incorrectly delete some triangle, we can always shut the project down without saving it and reopen.

- Choose the Select Triangle tool from the toolbar.

- Holding the Ctrl key, multi-select several of the triangles at the downstream boundary by clicking the mouse on one triangle and dragging the cursor through the other triangles as show in the Figure 9.

- Press the Delete key to delete the highlighted triangles. Click yes to confirm the deletion.

- Make sure to zoom in and delete all the unnecessary triangles.

- The artificial lip should now be removed and the model will be able to appropriately channel the flow.

- SAVE Project .

Figure 9: A) The entire Lakina River TIN with an inset box showing were Figure 9B &C are located. B) The arrows shows which triangles to remove in order to eliminate the artificial lip. C) The zoomed in portion of the TIN after the triangles have been removed.
3. Breaklines

The triangles of the TIN created by SMS honor the Delaunay criteria which results in the creation of triangles as close to equilateral as possible. Since our dataset is composed of cross sections that are straight lines, the triangles connect a point on one section to the closest point on an adjacent section. To give a truer representation of the geometry, it is usually best to connect points of constant elevation.

A breakline is a feature line or polyline representing a stream channel, ridge or some other feature to preserve in a TIN. In other words, a breakline is a series of edges that the triangles should conform to. SMS includes a function to force breaklines into a TIN.

To see if the TIN needs to be adjusted using breaklines, the contour map will be evaluated.

- Select the Display Options Shortcut tool, make sure Scatter is highlighted, then select All Off at the bottom and turn on Contours.

- Under the Contour Tab, make sure the Contour method is Color Fill and Linear in the drop down menu. Then click OK and zoom to the Lakina Scatter.

- To better visualize the elevations, the contour display range can be changed. Select the Contour Options shortcut at the bottom of the screen. In the Data Range options select Specify a range. For Min enter “1360” and for Max enter “1400”. Change the transparency to “60%”. The display should appear something like Figure 10.

- Zoom to Scatter.
For the most part, the Lakina River contours follow the stream channels in the aerial photo. But for a learning experience we will add some breaklines and modify the TIN.

- Turn off the Lakina Scatter in the Project Explorer.

Notice the two whitish diagonal channels upstream of the bridge Figure 11A. Refer back to Figure 10 and notice the contour lines do not incorporate those smaller side channels. Breaklines will be added so they are:

- Turn the Lakina Scatter layer back on.

- Select the Display Options Shortcut tool, make sure Scatter is highlighted, then turn on Points, Triangles and Breaklines and turn off the Contours. Click OK to exit.

- To add breaklines, we will use the Create Scatter Breaklines tool from the toolbar.
After selecting the Create Scatter Breaklines tool, we will draw in breaklines similar to Figure 11B. Start by clicking on a scatter point and work your way down the white channel, clicking on additional points in the subsequent cross sections. Ideally, the points would have similar elevations (contour color). The similar elevation concept does not always work, because we are decreasing in elevation as we work our way downstream. Regardless, continue along the channel clicking on points until you stretch the length of the channel. Double-click to end the breakline.

Repeat the process creating multiple breaklines to ensure a smoother channel bottom. Your breaklines should be created similar to those in Figure 11B.

Now the breaklines need to be forced into the TIN.

To do this select Breaklines | Force Breakline from the menu. The dataset will be re-triangulated and will now appear similar to Figure 12, which now has contours that incorporate those smaller side channels.

SAVE Project.
4. Conclusion

In summary, the TIN was cleaned up by removing an artificial downstream lip by manually removing triangles. Additionally, breaklines were added to improve the elevation data representation.
Modeling Exercise #3 – Parameters

Building Feature Objects

1. Introduction

This modeling exercise describes the process of creating a model domain using the Map Module and the conceptual modeling approach. This approach is a powerful option to generate a model using feature objects to define parameters for boundary conditions, materials, and automatic mesh generations.

2. Initializing the SMS Workspace

The previous Background Data modeling exercises showed how to import a scatter set of elevation data and work with images in SMS. The data from the Background Data exercise will be used as the starting point for this workshop. To ensure consistency, read in a set of completed files from the end of the previous exercise. To do this:

- Open SMS and select Open.
- Navigate to “SMS Modeling Exercises\Exercise 2 – Background Data B” directory.
- Select the file named “Lakina River Editing.sms.”, then select Open.

If it asks “Do you want to delete existing data?,” Click Yes.

3. Using Feature Objects

The purpose of the exercise is to become familiar with creating feature objects in SMS, and to create the components (coverages) needed to run an SRH-2D model simulation. The background image and scatter set will be used as a guide to create the feature objects. Feature objects in SMS include points, nodes, arcs, and polygons.

Usually for an SRH-2D simulations, a Mesh Generator coverage, a Boundary Condition coverage, and a Materials coverage are required. An optional Monitor Points coverage is also usually defend. To practice creating feature objects in SMS, one of each of these coverages will be created.

4. Mesh Generator Coverage

First, a Mesh Generator coverage will be created to define the model domain. This coverage will also be used in the next lesson to assign the automatic mesh generation.
• R-Click on the “Area Property” coverage in the Project Explorer.

• Select Type | Generic | Mesh Generator.

• R-Click on the “Area Property” coverage again and select Rename.

• Rename to “MeshGen”.

4.1. Creating Feature Arcs

Feature arcs are used to define boundaries for the model. Feature arcs are often digitized directly inside SMS using the background data as a guide. They may be created from other data imported into SMS such as scatter, CAD or GIS data.

To define the boundary “model extent” using feature arcs:

• Let get the display set up: Select the Display Options Shortcut tool, make sure Scatter is highlighted, the click All Off and select Contours.

• Click on the Contours tab, change the Contour method to Color Fill and Linear and change the Data range Max to 1400. Click OK to exit.

• Click “MeshGen” coverage in the Project Explorer.

• Choose the Create Feature Arc tool from the Toolbar.

• Draw a boundary within the contour define. Repeat, the boundary must be completely within the TIN. Start by clicking on the map, then click along the boundary to define the model domain. Don’t worry about uniform spacing, just quickly click out the general boundary similar to Figure 13. The forested hill (Red Area) does not need to be completely included. Double-click on your starting point to finish the arc boundary.

• To turn on the Nodes, Arc and Vertices, select the Display Options Shortcut tool, make sure Map is highlighted, then select Node and Arc.
  
  Select the Line Attributes and make the arc width “5” and the color black | Ok | OK. Your map should appear something like Figure 13.

• SAVE Project.
4.2. Adding Internal Features

Internal features such as river banks, ridges, structures, roads, material zones, and boundary conditions are defined using arcs and polygons. We will now add additional arcs to define internal features.

- Choose the Create Feature Arc tool from the Toolbox.
- Make an Arc in the shape of both the roads as shown in Figure 14. You can end the arc by double-clicking on the boundary arc.
Next we will draw arc “circles” around the Bridge Piers:

- Un-select the Lakina Scatter coverage and select the Substructures coverage in the Project Explorer.

- Turn on the Substructures points by selecting the Display Options Shortcut tool, make sure Scatter is highlighted, then select Points and un-select Contours.

- Zoom in to the bridge, click on the “MeshGen” coverage and using the Create Feature Arc tool to draw little circles around each of the six piers (see Figure 15).

This is the Cookie Cutter approach for piers, as we are basically cutting out the mesh where the piers are located. The model will treat the boundary conditions as a “Wall” (No flow, no slip).

- SAVE Project.
4.3. **Redistribute Vertices**

The primary function of the vertices of an arc is to define the geometry of the arc. If the arcs are to be used for automatic mesh generation, the spacing of the vertices is important. The spacing of the vertices defines the density of the elements in the resulting mesh. Each edge defined by a pair of vertices becomes the edge of an element. The mesh gradation is controlled by defining closely spaced vertices in regions where the mesh is to be dense and widely spaced vertices in regions where the mesh is to be coarse. To redistribute the vertices:

- **Zoom to coverage by R-clicking on MeshGen and selecting Zoom to Coverage.**
- **Select the Select Feature Arc tool, then select Edit in the main menu and Select All.** The arcs should now all be highlighted.
- **R-click on any arc and select Redistribute Vertices... to open the Redistribute Vertices dialog.**
- **Change the Average spacing to “15.0” feet and click OK to redistribute the vertices.**
- **To make the vertices more visible, select the Display Options Shortcut tool, make sure Map is highlighted, then using the Symbol Attributes for Vertex, change the size to 10. Click Ok | OK to get back to the map.**

Let’s add more vertices to the pier feature arcs.

- **Zoom into the piers and using Select Feature Arc tool draw a box around the 6 piers. R-click one of the highlighted arcs and select Redistribute Vertices... to open the Redistribute Vertices dialog.**
- **Change the Average spacing to “2.0” feet. Click the OK button to redistribute the vertices.**
- **SAVE Project.**

4.4. **Building Polygons from Arcs**

Next polygons must be generated. Before creating polygons, the data should be cleaned to avoid potential errors. To clean the feature arcs:

- **Zoom to coverage by R-clicking on MeshGen and selecting Zoom to Coverage.**
- **Select Feature Objects in the main menu and then select Clean... to open the Clean Options dialog.**
• Turn on **Snap nodes and vertices** and set the **Tolerance** to “0.001.” This means that if two feature nodes are within 0.001 feet of each other, they should be merged into a single point.

• Turn on **Intersect arcs**.

• **Make sure Remove dangling arcs** is **un-selected**, otherwise it will remove the 6 pier arcs.

• Click the OK button to clean the data.

Next we will build polygons from the cleaned arcs.

• **Be sure no arcs are selected**, then choose **Feature Objects** in the main menu, then select **Build Polygons**.

• To confirm the polygons have been created, choose the **Select Polygon** tool from the toolbar. Click on any polygon to select. The selected polygon will be highlighted.

• **SAVE Project**.

This Mesh Generation coverage will form the basis of the automatic mesh generation. We will stop the process for the mesh generation coverage at this point. These polygons will need to be subdivided and mesh generation parameters need to be assigned or edited.

5. **Boundary Conditions Coverage**

A Boundary Conditions (BC) coverage will be created to define the inflow and outflow for the simulations. For this exercise, the inflow and outflow boundaries will be specified and monitors lines will be created.

• R-Click on “Map Data” in the Project Explorer and select **New Coverage**.

• For the coverage type select the SRH-2D | **Boundary Conditions** coverage and specify the coverage name as “BC.”

• Select OK and a new SRH-2D boundary conditions coverage will be created.

• **SAVE Project**.

5.1. **Define Monitor Line Arcs**

Monitor lines can be created to extract flow information from the model. Two monitor lines will be created for this model to check flow continuity on either side of the bridge as shown in Figure 16.

• Create the monitor line arcs up-stream and down-stream of the bridge.
• Click on the “BC” coverage in the Project Explorer to active the coverage.

• We need to first make the existing arcs from the “MeshGen” coverage visible, so select the Display Options shortcut tool, make sure Map is highlighted, then select Inactive coverage option. This will make the arcs from the inactive coverage visible. Click OK to exit.

• Choose the Create Feature Arc tool. Draw in the lines similar to what is seen in Figure 16.

• SAVE Project.

The default boundary condition type is Monitor Line, therefor the two arcs just created are already defined as monitor lines and do not need to be reassigned. In the next steps, two additional arcs will be created for the inflow and outflow boundary conditions.
An arc defining the inflow boundary will now be created.

- Inflow and Outflow boundary conditions must lie along the boundary of the model domain. We will use the arcs from the inactive coverage for tracing.

- To accurately trace features in an inactive coverage, the snapping feature will need to be activated. Select *Edit*, then *Preferences...* and choose the “Map” tab. Ensure that “Snap feature objects to displayed inactive coverage nodes and vertices” is turned on.

- To create the inflow arc, we will using the *Create Feature Arc* tool. Begin the arc by clicking on the node in the upper right corner of the domain (see Figure 17). With the snapping feature turned on, SMS will snap to the inactive node. Draw the line across the upstream boundary by clicking on any nodes or vertices along the inactive northeastern boundary. Terminate the arc by double clicking.

![Figure 17: Completed BC coverage arcs.](image-url)
• The newly created “Monitor-Line” needs to be converted into an inflow boundary arc. Select the **Select Feature Arc** tool and double-click on the arc to launch the **Linear BC** attributes dialog. Select **Inlet-Q** (subcritical inflow) in the drop down menu.

• The inflow will be defined as Constant. Enter 1285 cfs in the field for the **Constant Q**. Click OK to close the dialog.

• SAVE Project.

5.3. **Define Outflow Arcs**

An arc defining the outflow boundary will now be created.

• The outflow arc will be created in the same manner the inflow arc was created. Using the **Create Feature Arc** tool draw a arc across the downstream boundary in the lower left corner of the domain (see Figure 17).

• Convert the arc to an outflow boundary arc. Select the **Select Feature Arc** tool and double-click on the new arc to launch the **Linear BC** attributes dialog. Select **Exit-H** (subcritical inflow) in the drop down menu.

• For this case the outflow will be defined as **Rating Curve** in the drop down menu for **Exit water surface Options**.

• Double-Click the “Undefined” box to open the XY Series Editor.

• Then select **Populate**. The Populate dialog will be used to generate a best estimate of the rating curve variables (Figure 18A). Fill in the form with the following information:
  
  o Type: Normal depth rating curve.
  o The model needs to know what scatter set to use for the **Ground Elevation Dataset**:
    - Select “z” under the Lakina Scatter Data | click OK.
  o Units: U.S. Units
  o Composite Mannings N: 0.035
  o Slope: 0.01 ft/ft
  o Populate Flows.
    - Min: 500
    - Max: 6000
    - Delta: 500
- Click “Add” to populate flows from 500 to 6000 cfs with an interval of 500 cfs.
- Click OK and the XY Series Editor should be populated with a Rating Curve (See Figure 18B).

![Figure 18: A) Rating curve population values. B) Rating curve.](image)

- Click OK to close the XY Series Editor. The “undefined” box should now have a red rating curve on it.
- Click OK to finish defining the outflow boundary condition.
- SAVE Project.

6. **Material Coverage**

A Material coverage will be created to define the unique material zones for the simulation. There are two ways to create a Materials coverage. It can be created manually by digitizing polygons to define the material zones or by reading polygons from a shape file.

6.1. **Digitizing a Materials Coverage**

One of the material zones has already been digitized with the roadway arcs were created with in the mesh generator coverage. To utilize this work, a duplicate of the mesh generator coverage will be used as a starting point.

- R-click on the “MeshGen” coverage in the Project Explorer and select Duplicate. A new coverage will be created with the a name “MashGen (2)”.
- R-click the “MashGen (2)” coverage and select Rename. Rename the coverage to “Materials”.

Page | 1-28
• R-click on the “Materials” coverage.

• Select Type | Models | SRH-2D | Materials.

First, names will be assigned for the materials to be used (roughness values will not be assigned at the point).

• Select the Edit | Materials Data menu command. The Material Properties dialog will appear.

• Initially only the “unassigned” material will be present in the table. This material can be used to assign areas where water will not flow. We will create three new materials and name them “Channel”, “Trees”, and “Roads”.

• Add a new material by clicking on the button. A new material called “new material” will be created.

• Double-click on the new material name and change it to “Channel”.

• Repeat the previous two sets to add material names for “Trees” and “Roads”.

• Double-click on the Color boxes to change their colors and textures.

• For each of the materials, leave the Manning’s Roughness as Constant, but change the Constant N values to:
  
  o Channel: 0.035
  o Trees: 0.1
  o Road: 0.015

• Click OK to exit the Material Properties dialog.

Arrows defining the material zone will now be created. The arcs already created defining the roads will be used to define the road material zone. Other polygons will be digitized to enclose other material zone areas.

• Un-select the “BC” coverage in the Project Explorer.

• Select “Material” in the Project Explorer.

• Choose the Create Feature Arc tool from the Toolbox.

• Quickly click out arcs to define polygons representing forested area (Trees). All other areas will be assumed to be the “Channel” (See Figure 19). Don’t spend too much time on this. It does not need to match the figure perfectly.
• Once complete, build polygons using the Edit | Select all | Feature Objects | Build Polygons menu command.

Next the material must be assigned to each polygon.

• Select the Display Options tool, make sure Map is highlighted, then select the Polygon: Fill and General: Legend option. Click OK to exit.

• To define the polygons choose the Select Polygon tool | Edit | Select All | R-click the “Materials” coverage in the Project Explorer and choose Material Properties to launch the Assign Material Properties dialog.
  
  o Because there are mostly “Trees” polygons we will select the “Trees” material. Make sure the Constant N is 0.1 and the texture represents trees (i.e. is green). Click OK to close the menu. All polygons should be the textures you choose for “Trees”.

Figure 19: Material coverage polygons.
• Now double-click on the polygon defining the “Channel”.
  
  o Select the “Channel” material. Make sure the Constant N is 0.035 and the texture represents a channel (i.e. is Blue). Click OK to close the menu.

• Repeat for the two road sections. Make sure the Constant N is 0.015.

![Figure 20: Defined material coverage polygons.](image)

• With the Select Polygon tool, draw a box around the 6 piers. R-click one of the piers Assign Material Properties. Select “unassigned”. Click OK to exit.

• The completed materials coverage line arcs should look similar to the image in Figure 20. The color and patterns may be different based on your definition.

7. **Monitor Points Coverage**

A Monitor Point (MP) coverage will now be created. Monitor points can be defined to extract data values at specified points. Also, during the execution of the model, the water level at tow monitor points will be displayed on a graph and updated at each hour of the simulation. This provides a good option for monitoring the solution during execution to determine that the solution has reached steady state conditions or that the inflow hydrograph for an unsteady run has propagated to the outflow boundary.
7.1. Creating the Monitor Points Coverage

- R-click on the “Map Data” item in the Project Explorer and select New Coverage. This will cause the New Coverage dialog to appear.

- Select Models | SRH-2D | Monitor Points coverage type. Leave the default name “Monitor Points”. Click OK.

![Monitor Points](image)

Figure 21: Monitor Points.

- Using the Create Feature Point tool, create monitor points as shown in Figure 21.

- SAVE Project.

8. Conclusion

This concludes the Parameters exercise. You should now be familiar with some of the features in SMS for creating a model domain, creating coverages and using feature objects to assign model parameters.
Modeling Exercise #4 – Geometry

Generating a Mesh

1. Introduction

This modeling exercise will provide an example of how to generate an unstructured grid (or mesh) of the Lakina River Bridge model domain suitable for use in a hydrodynamic simulation in SMS. The workshop includes:

- Subdividing feature polygons in a Mesh Generator coverage.
- Adding automated mesh generation attributes to feature polygons.
- Specifying material and geometric data sources for polygons.
- Generating a mesh.

We will begin with the data from the previous exercise. To ensure consistency, read in a set of completed files from the end of the previous exercise. To do this:

- Open SMS and select Open.
- Navigate to “SMS Modeling Exercises\Exercise 3 – Parameters” directory.
- Select the file named “Lakina River Parameters.sms.”, then select Open.

If it asks “Do you want to delete existing data?,” Click Yes. Select No, if asked if you would like to generate image pyramids.

2. Subdividing the Domain

Some models can be defined using a single polygon in the Mesh generator coverage and creating triangles through the entire domain. For most, however, the model domain will be broken up into several polygons. For this model, a polygon defining the roads have already been defined. Polygons defining the piers have been defined too.

3. Model Resolution

The model resolution in each area of the mesh is controlled by the spacing of the vertices on the polygon arcs. Vertex spacing can be reduced to create a fine grid in critical areas and a coarser grid in less critical areas. Care must be taken to ensure that the transition between high and low resolution areas is gradual. For this model, we already redistributed the vertices using a 15 foot spacing for all the arcs, then further redistributed the pier polygon spacing to 2 foot spacing.
4. **Meshing Parameters**

Meshing parameters will be defined for each polygon in the domain to define how elements will be automatically generated in the mesh. Users can experiment with variations of mesh resolution by changing parameters or vertex distribution of individual arcs and polygons. These changes can be visualized before actually creating the mesh. The mesh is automatically created using a single command.

During the automated meshing process, SMS interpolates elevation for each node from a background data source (scatter set). SMS also assigns a material type to each element created. Material types are specified using a separate SRH-2D Materials coverage.

This example will illustrate various options for mesh generation parameters assigned to feature polygons. Variation in mesh type and various ways to adjust vertex spacing will be explored.

4.1. **2D Mesh Polygon Properties Dialog**

- Un-select the “BC”, “Materials” and “Monitor Points” coverages in the Project Explorer.

- Click on the “MeshGen” coverage in the Project Explorer to active the layer, then R-click on it and select Zoom to Coverage.

- Choose the Select Feature Polygon tool. Select Edit in the main menu, then choose Select All.

- Choose Feature Objects in the main menu, then click on Attributes to open the 2D Mesh Multiple Polygon Properties dialog.
  - Select Mesh type and in the drop down menu choose None. We are starting with None because we have 6 pier polygons that will have no mesh and only 3 polygons (two roads and the main domain) that will have meshes.
  - Select Bathymetry type and in the drop down menu choose Scatter Set. We are choosing to do this because the three meshes will use the scatter set.
    - Click on Scatter Options… then under the Scatter Set to Interpolate From select the “z” under Lakina Scatter (active).
  - Click OK | OK.

- Select the Display Options shortcut tool and make sure Map is highlighted. Select Polygon: Fill if it is not already. You can change the color for None if you want.
4.2. Paving

Paving meshes work for all polygons shapes. The paving method fills the polygon with triangles. The size and number of elements created inside the polygon is based on the vertex spacing of the polygon arcs and the bias specified by the user. The paving method creates rows of elements working from the polygon boundaries to the center.

- Choose the Select Feature Polygon tool and double click on the main polygon to open the 2D Mesh Polygon Properties dialog (Figure 22).

![Figure 22 Mesh Polygon Properties dialog.](image)

- We already specified the Mesh Type to be None and the Bathymetry Type to be Scatter Set.
  - Change the Mesh Type to Paving using the drop down menu.
- Click the Preview Mesh button. SMS updates the preview to show the polygon shape and how that shape will be filled with elements using the current settings.
- Beneath the preview window, the dialog includes a toolbox for viewing and modifying the arcs of a polygon without exiting the dialog. Select the Zoom tool to the Toolbox dialog. Drag a box around the bridge area (piers) of the polygon (Figure 23).
• Select the *Pan* tool. Click on the dialog display window and drag the mouse to move around the polygon.

• Press OK to close the Mesh Polygon Properties dialog.

Figure 23: Paving mesh preview around the piers.

• SAVE Project.

4.3. **Patches**

The “Patch” mesh type is generally used to define areas of the mesh that can be represented by 4 “sides” made up of one or more arcs. As with paving, the patch mesh type uses the vertex distribution along the sides to set the element size. For patches, elements are created by “connecting” the vertices on opposite sides in a “checkerboard” pattern. If the number of vertices on opposite sides is equal, then the patch will contain only quadrilateral elements. If the numbers of vertices is not equal, triangles are inserted as transition elements. Figure 24 shows an example of meshes created using the patch method for both cases.

Figure 24: Patching with equal and unequal number of vertices on opposite sides.

Patches often create better mesh representation for long thin polygons. Therefore, patches will be used to represent the road sections of the mesh.
• Choose the Select Feature Polygon tool and double click on the western road polygons to open the 2D Mesh Polygon Properties dialog.

• Change the Mesh Type to Patch using the drop down menu.
  
  o An error should pop up: “Patches require three or four edges. Select a node and set the Node Options to Merge to treat multiple feature arcs as a single edge”.

  o Click OK to exit.

Before we can choose a Patch mesh type, we need to add some Nodes and change the distribution of vertices, so the polygon has 4 sides and equal vertices on opposite sides. Currently there are only 2 nodes, which are the blue dots in Figure 25. To add Nodes we must exit out of the Mesh Polygon Properties dialog.

• Click OK to exit the dialog.

• Select the Zoom tool and zoom into the western road polygon (See Figure 26).

• Select the Select Feature Vertex tool from the Toolbar. Click on one of the vertices represented with a black star in Figure 26. R-click on the highlighted vertex and choose Covert to Node. The vertex should now be a node and probably turned white in color.

• Repeat this process for the second black started vertex in Figure 26.

Figure 25: Road section polygon mesh preview with two Nodes.

Figure 25: Road section polygon mesh preview with two Nodes.
Figure 26: Wester road polygon. The stars indicated the verities that need to be covered to nodes.

Now there are 4 sides to the road polygon, so the Mesh Type can be changed to Patch.

- Choose the Select Feature Polygon tool and double click on the western road polygons to open the 2D Mesh Polygon Properties dialog. There should now be four blue nodes in the polygon mesh preview.

- Change the Mesh Type to Patch using the drop down menu. Click on Preview Mesh.
  - Select the Zoom tool and zoom in to see if there are any triangles. For the most part the Patching looks good, but there are a few triangles (Figure 27A).

Figure 27: Mesh Patching. A) The number of vertices is not equal, triangles are inserted as transition elements. B) The number of vertices is equal and there is a complete “checkerboard” pattern.

- To make the vertices even on opposing sides of the polygons select the Select Feature Arc tool from the Toolbox. Select the arc on the top of the polygon. Selecting an arc activates the Arc Options section in the lower left of the dialog. Select the Distribute option. Change the number of vertices to 34.

- Now select the bottom polygon arc using the Select Feature Arc tool. Select the Distribute option and change the number of vertices to 34, so opposite side have the same value.

- Click Preview Mesh. The triangles should no long be there and the whole polygon should be a “checkerboard” pattern (Figure 27B).
This process needs to be repeated for the eastern road polygon.

- Click OK to exit the dialog.
- Select the Zoom tool and zoom into the eastern road polygon (See Figure 28).
- Select the Select Feature Vertex tool from the Toolbar. Individually select and convert the 3 vertices represented with black stars in Figure 28 to nodes.

![Eastern road polygon. The stars indicated the vertices that need to be covered to nodes.](image)

- Choose the Select Feature Polygon tool and double click on the eastern road polygons. Note, that for this polygon, the bottom “side” of this polygon has two arcs.
- Select Patch in the Mesh Type drop down menu.
  - The error message appear again. “Patches require three or four edges. Select a node and set the Node Options to Merge to treat multiple feature arcs as a single edge”.
- To correct this, the middle blue node along the bottom side must be specified as a vertex for this polygon in order to merge the arcs. Click OK to exit the error message.
- Select the Select Feature Point tool from the Toolbox. Select the middle blue node on the bottom arc.
  - The Node Options section of the dialog window will now be active. Select Merge in the drop down menu. Select again anywhere in the open space in the dialog display window. Note that the node has changed from blue to red indicating the node is a Merge node.
  - Select Patch in the Mesh Type drop down menu. Click the Preview Mesh button.
• Select the **Zoom** tool and zoom in to see if there are any triangles. If there are triangles, make the distribute vertices values equal on opposing arcs. Notice that the Bathymetry is already set to Scatter Set, which is what we want.

• Click OK to exit the dialog.

All of the polygons have now been assigned a *Mesh Type*. Make sure the “Piers” have *No Mesh*, the “Roads” have *Patch Mesh*, and the rest of the “Domain” has *Paving Mesh*. This can be easily accomplished by making sure the polygon colors match the legend (See Figure 29. Your colors might be different.

• **SAVE Project**.

![Figure 29: Assigned Mesh Types and Legend. A zoomed in view of the piers and the end of the eastern road.](image)

5. **Assigning Materials**

One final polygon attribute to discuss is the assignment of materials data. For SRH-2D, a separate SRH-2D Materials coverage defines the materials for each mesh node. The option in the dialog to specify materials for feature polygons in this coverage is dimmed and is not available for SRH-2D.

6. **Generating the Mesh**

Until now, only a preview of the meshing of individual polygons has been viewed. The entire mesh will now be created using the “instructions” stored as parameters for each polygon:

• **R-click** on the “MeshGen” coverage in the *Project Explorer* and select **Zoom to Coverage**.
• Make sure no polygons are selected.

• R-click on the “MeshGen” coverage and choose the command Covert \ Map \ 2D Mesh. The 2D Mesh Options dialog appears.

• Leave the default setting and click OK.

• Specify the Mesh Name as “Lakina Mesh” and click OK. SMS generates the elements of the mesh and interpolates the elevations to the nodes.

• To view the mesh, un-selected \ Map Data\ in the Project Explorer and click on “Lakina Mesh” coverage to active the layer.

• Select the Display Options tool, make sure 2D Mesh is highlighted. Be sure Nodes and Contours is un-selected and Elements is selected. Click OK to exit.

• Toggle off the Lakin_13.tif image to better view the mesh.

• Figure 30 shows how the mesh may look.

• SAVE Project.

Figure 30: Lakina River Mesh.
7. **Evaluating the Mesh**

After creating a mesh, it is a good idea to evaluate the mesh for quality and to be sure that it was created as intended. Two separate checks should be made in reviewing meshes.

7.1. **Mesh Quality**

First visually inspect the mesh to be sure that the elements look reasonable and that they were created as desired. SMS also provides an option to detect areas which are outside the mesh quality guidelines. These options may not identify all problems, nor do all identified issues need to be corrected. However, they will show potential problem areas for the mesh.

The *Mesh Quality* options in SMS display potential problems with individual elements. To turn on the *Mesh Quality* options:

- Select the *Display Options* tool and make sure *2D Mesh* is highlighted.
- Toggle on the *Mesh quality* option.
  - Click the Options button to the right of the *Mesh quality* option.
  - The *Element Quality Checks* dialog controls the thresholds of what is defined as a good or a bad element. Turn off the *Maximum slope* option. This generally only applies to model limited to subcritical flow such as old RMA2 model.
  - Change the *Maximum interior angle* to 110 degrees.
  - Click the OK twice to exit both dialogs.

Elements that violate the specified criteria are now highlighted with a color for each mesh quality option. The most common problem are area change and element interior angles. Once SMS has highlighted potential problems, it is up to the user to either fix or ignore the warning.

*There are no hard and fast rules when it comes to element quality. The real measure is what solutions the engine can generate for the given mesh. For this case some issues occur. These issues could be addressed by adjusting the meshing parameters and vertex spacing on the individual polygons.*

7.2. **Elevation contours**

It is a good idea to review the elevations for the mesh to be sure that the mesh generated reasonable represents the underlying background elevation data.

- Select the *Display Options* tool and make sure *2D Mesh* is highlighted.
• Turn off the Mesh quality option and turn on the Contours.
  
  o Select the Contours tab and be sure Color Fill is selected. Click the Color Ramp button and select Reverse if necessary to display the lower elevations as blue. Click OK. Un-select Specify a range and make the Transparency “0”.

• Click OK to exit the dialog. Color elevation contours will be displayed.

Check the contours to be sure the elevations are properly represented. If a polygon is not with the underlying scatter set, zero elevations will be assigned and will be evident. It will be clear if some polygons were not assigned to interpolate from the scatter set.

• Select the Zoom tool and zoom into the piers to make sure no elevation contours (color) exist.

• SAVE Project.

8. Conclusion

This concludes the Geometry modeling exercise. This exercise has illustrated some of the features SMS provides for mesh generation for a model domain definition.
Modeling Exercise #5 – Model Simulation

SRH-2D Simulation Setup and Model Run

1. Introduction

All the components needed for the SRH-2D model have now been created. These include a mesh, boundary conditions, materials and a monitor points.

We will begin with the data from the previous exercise. To ensure consistency, read in a set of completed files from the end of the previous exercise. To do this:

- Open SMS and select Open.
- Navigate to “SMS Modeling Exercises\Exercise 4 – Generating a Mesh” directory.
- Select the file named “Lakina River Mesh.sms.”, then select Open.

If it asks “Do you want to delete existing data?,” Click Yes. Select No, if asked if you would like to generate image pyramids.

2. Creating a Simulation

An SRH-2D simulation will now be created:

- R-click in the blank area at the bottom of the Project Explorer and select New Simulation | SRH-2D.
- R-click the “Sim” coverage and select Rename. Rename the coverage to “Q2”.

3. Linking Components

Links will now be created to assign the components to the simulation. Typically for an SRH-2D simulation, a mesh will be linked to define the geometry. Coverages (layers) will also be linked which specify the boundary conditions, the materials and monitor points for the simulation. Because simulations contain links to SMS objects, objects can be shared among multiple simulations.

- Create a link to the mesh by dragging the mesh named “Lakina Mesh” in the Project Explorer onto the simulation “Q2”. This is done by clicking on the mesh item in the Project Explorer and dragging it underneath the simulation. A heavy black line will appear to the right place as shown in Figure 31A.
Next add boundary conditions, materials and monitor points coverages to the simulation. Click and drag the “BC”, “Materials”, and “Monitor Points” coverages into the simulation as was done with the mesh. No other item will be linked in the simulation. When complete the “Simulation Data” in the Project Explorer should appear as show in Figure 31B.

Figure 31: A) Project Explorer illustrating heavy black line when dragging and dropping the mesh. B) Linked coverages.

As shown above, the components in the simulation have a small arrow next to the icon indicating these objects are just links and not the actual object. Therefore when the objects are modified, the simulation links back to the updated object. When components are modified they do not need to be “relinked” (dragged) to the simulation.

4. Define Model Control

Then final step before running the model is to specify the model control parameters. In the model control, we specify the solution type, run time, time step, initial conditions, turbulence options and output options. In this section we will review and set each of the model control parameters and options. To do this:

- R-click on the “Q2” simulation in the Project Explorer and select Model Control to launch the Model Control dialog. There are three tabs in this dialog: General, Flow, and Output.

4.1. General Tab
• For **Simulation Description** enter “2 year flood”. This is a descriptive text only that will appear in the output file.

• For **Case Name** enter “Q2”. This name will be used for the prefix for the results file names in the folder where the file results are stored.

• Be sure the checkbox for **Temperature Modeling** is not checked.

• For **Start Time** leave the value at the default of 0 hours.

• For **Time Step** specify 1 second. The timestep is the most critical parameter affecting model stability. A rule of thumb is that time step should be set such that water does not move across more than one element in a single timestep. It is therefore dependent on both water velocity and element resolution. Experience is required to determine appropriate time steps. Usually a timestep between 1 and 10 seconds is specified, and can be reduced if the model becomes unstable.

• For **Total Simulation Time** specify 1.25 hour. A time should be specified which allows for the entire hydrograph to propagate to the downstream boundary for an unsteady run, or for the simulated steady state flow to reach equilibrium at the downstream boundary. Results and monitor points can be used to verify these conditions. Be sure that the flow values specified in the table for the boundary conditions coverage cover enough time for the **Total Simulation Time**.

• Specify **Dry** in the drop down box for **Initial Condition**. This option defines how the initial water level is set for running the model. Most models are run beginning with a dry domain.

4.2. **Flow Tab**

• Select the **Flow** tab.

• Leave the default turbulence parameters in the **Flow** tab. SRH-2D uses a global turbulence model with a default **Parabolic** turbulence of 0.7.

4.3. **Output Tab**

• In the **Output** tab, make sure the **Results Output Unit** is set to English, and specify the **Results Output Frequency** as 0.0167 hours.

• Click OK to exit the **Model Control** dialog.

• SAVE Project.

5. **Running the Model**

The SRH-2D model simulation is now ready to run. To do so:
• R-click on the simulation “Q2” and select *Save, Export and Launch SRH-2D.*

• A message may appear notifying you that the BC and Materials coverage will be renumbered. Click OK.

*SMS launches the model wrapper and displays the screen output for the numerical model.* SRH-Pre requires only a second or two to run. Once it has completed SMS will automatically launch SRH-2D. Once the SRH-2D run begins, you will see a model wrapper consisting of three DOS windows. The first is simply a title graphic for SRH-2D including some version information. This window can be minimized. The second window is Residual Monitor. This plots the changes in residuals for the solution procedure. It can give some indication of model stability but mostly will indicate how far the model is progressing. The third window plots the changes in water levels for the monitor points. This can be used to see changes at the inflow and outflow locations and for this model can give an indication that the model has reached steady state.

• Move and resize such that the second and third windows are visible as shown in Figure 32.

![Figure 32: SRH-2D Model Wrapper](image)

If the model successfully runs to completion, SRH-2D will display the dialog below.

![Program Terminated with exit code 0 Exit Window?](image)

• Click *Yes* to close the SRH-2D model wrapper.

• Click *Exit* to close the SMS model wrapper.
During the model run, SRH-2D writes out several files in the same folder where the SRH-2D model inputs are written out. Most of those files are Restart files written out at each timestep. The model results are stored in a ZMDF file which can be loaded into SMS for further visualization. The model results should automatically be read in.

6. Post-Processing

When SMS finishes reading the solution files, several datasets are added under the Mesh Data folder in the Project Explorer. The solutions datasets include computed depths and velocities, water surface elevations, Froude number for each node and each time step in the mesh. Contour and vector display of these datasets can be generated.

To update the display settings for dataset viewing:

- Select the Display Options tool and make sure 2D Mesh is highlighted.

- Click the All Off button at the bottom of the window and then select Mesh boundary, Contours, and Vectors.

- Click on the Contours tab:
  - Make sure the Contour method is Color Fill.
  - Select Specify a range and enter a Min: 0.1 and Max: 10.
  - Un-select the Fill below and Fill above.

- Click on the Vectors tab.
  - In the Vector Display Placement and Filter section, select on a grid from the drop-down menu.
  - For the Origin select Relative to bed in the drop-down box and enter 10.0 for the Offset.

- Click OK to exit the Display Options dialog.

- Select the Lakina_13.tif image to turn it on.

- Click on the Water_Depth_ft dataset in the Project Explorer to activate it.

- Below the Project Explorer is a Timeset window. Select the initial time set (i.e. 00:00:01). Since the project started dry, there will be little if any water shown flowing into the mesh at the upstream boundary. Use the arrow keys on the keyboard to scroll down through the time steps to observe the flow into the model domain over time (Figure 33).
• Leaving the solution at the last time step, alternately select the Water Depth, Velocity magnitude, and Froude number datasets in the Project Explore.

• Continue to explore the solution datasets by changing display options as desired.

• SAVE Project.

Figure 33: Water depth 3 minutes into the simulation.

7. Map Export

Once a desired map display has been created, it can easily be exported:

• Select File | Save As.

• Enter a name of your choice.

• Change the Save as type to JPEG Image Files (*.jpg).

• Click Save.

• The model then exports an image file that can be opened in other image viewers.
8. **Create Profile Plots**

It is often useful to create profile plots of the datasets to represent or show the data. This is done using arcs in an observation coverage in the *Map* module.

- R-click on the “Map Data” and create a new Observation type coverage named “Observation”.
- Select the new observation coverage in the *Project Explorer*.
- Using the *Create Feature Arc* tool create an arc down the center of the reach from the downstream boundary to the upstream boundary.
- Select *Display | Plot Wizard* menu command to launch the *Plot Wizard*, then select the *Observation Profile* option. Click *Next*.
- Be sure Arc 1 is checked in the *Coverage* sections.
- Under *Datasets* select *Specified*. Select the elevation and water elevation datasets.
- Leave the *Time steps* option as *Active* and click *Finish*. A profile plot of the water surface elevation along with the ground elevation will be displayed (Figure 34).

![Figure 34: A profile plot of the water surface elevation along with the ground elevation.](image)

9. **Create Animation**

A film loop (animation can be created showing a flow trace of the solutions:

- R-click on *Lakina Mesh | Zoom to Mesh*. This will also activate the *Mesh* menu options.
- Select *Data | Film Loop* menu command.
• Check Create AVI File option, then click on the browse button. Specify a name of "LakinaRiver" for the filename ofr the AVI file. Click Save.

• Select the Flow Trace option, then click Next.

• Click Next | Next | Finish to accept the default Display Options. A flow trace animation will be created and should run in the Play AVI Application (Figure 35). This animation will be saved in a file named as specified and can be played in a Powerpoint presentation or other media player.

• Zoom into a specific area and create a new animation. Experiment with the different options for creating animations by creating more versions.

Figure 35: Lakina River Film Loop.

10. Conclusion

This workshop provided a basic introduction to setting up and running an SRH-2D simulation. Different post processing methods were also evaluated.
Day 2

2D Hydraulic Modeling of Rivers using HEC RAS-2D
HEC-RAS Table of Contents

Modeling Exercise #1 – ArcMap ................................................................................................................ 2-1

Developing a Terrain Model ..................................................................................................................... 2-1

1. Introduction ............................................................................................................................................ 2-1

2. Developing a Terrain Model ............................................................................................................... 2-1

3. ArcGIS .................................................................................................................................................. 2-1

4. Setting the projection .......................................................................................................................... 2-1

5. Background Image .............................................................................................................................. 2-2

6. Digital Elevation Data ......................................................................................................................... 2-2

7. Adding Pier Locations into Terrain Data using ArcMap ........................................................................ 2-5

8. Create TIN ........................................................................................................................................... 2-7

9. Editing TIN .......................................................................................................................................... 2-9

10. Create Raster from TIN ...................................................................................................................... 2-11

11. Conclusion ........................................................................................................................................ 2-13

Modeling Exercise #2 – Background Data .............................................................................................. 2-14

Gathering Data .......................................................................................................................................... 2-14

1. Introduction ......................................................................................................................................... 2-14

2. Setting the Spatial Reference Projection ............................................................................................ 2-15

3. Loading the Terrain Model ............................................................................................................... 2-15

4. Background Image .............................................................................................................................. 2-17

5. Conclusion .......................................................................................................................................... 2-17

Modeling Exercise #3 – Development of a 2D Model ............................................................................. 2-18

Development of the 2D Computational Mesh ........................................................................................ 2-18

1. Introduction ......................................................................................................................................... 2-18
2. Drawing a Polygon Bounder for the 2D area .................................................................2-18
3. Adding Break Lines inside of the 2D Flow Area ............................................................2-20
4. Creating a Spatially Varied Manning’s Roughness Layer ...............................................2-21
5. Creating the 2D computation Mesh ................................................................................2-23
6. Potential Mesh Generation Problems .............................................................................2-26
7. Running the 2D Geometric Preprocessor ......................................................................2-28
8. External 2D flow area Boundary Conditions ................................................................2-28

Modeling Exercise #4 – Unsteady Flow Model..................................................................2-33

Unsteady Flow Simulation .................................................................................................2-33
1. Introduction ..................................................................................................................2-33
2. Performing the Computations ......................................................................................2-33
3. Viewing Output using RAS Mapper .............................................................................2-34
4. Creating Static (Stored) Maps ....................................................................................2-38
5. Time Series Output Plots and Tables ..........................................................................2-39
6. Conclusion ....................................................................................................................2-39
HEC has added the ability to perform two-dimensional (2D) hydrodynamic routing within the unsteady flow analysis portion of HEC-RAS. Users can now perform one-dimensional (1D) unsteady-flow modeling, two-dimensional (2D) unsteady-flow modeling (Saint Venant equations or Diffusion Wave equations), as well as combined 1D and 2D unsteady-flow routing. The 2D flow areas in HEC-RAS can be used in number of ways. The following are exercises based on the same Lakina River dataset that was used in the SMS workshop. The idea is to develop two different hydrological models using the same data, so a comparison can be made.

### Developing a Terrain Model

It is absolutely essential to have a detailed and accurate terrain model in order to create a detailed and accurate hydraulics model. The quality of the terrain data can be a limiting factor in the quality of the hydraulics model the user can create. Terrain data comes from many different sources, formats, and levels of detail. Currently HEC-RAS uses gridded data for terrain modeling. It is up to the user to gather data from multiple sources, create a good terrain model, then convert/export it into a gridded data format that can be read in by HEC-RAS.

It is necessary to create a terrain model before the user can perform any HEC-RAS model computations that contain 2D flow areas. This section of the workshop describes how to create a terrain model in ArcGIS. For details on creating terrain models with HEC-RAS Mapper, please review the chapter on HEC-RAS Mapper in the HEC-RAS User’s manual.

### ArcGIS

Because your computers are not equipped with ArcMap, set back and watch along for the remainder of the ArcMap exercise.

- Launch ArcMap from the desktop icon, start menu or from file browser. Select New Map | Blank Map.
- Save file: File | Save As and name it Lakina River Workshop.

### Setting the projection

The instruction below describe how to set the projection.
• **View** | *Data Frame Properties to open the Data Frame Properties dialog.*

• **Click on Projected Coordinate System | State Plane | NAD1983 (2011)(US Feet) | NAD_1983_2011_StatePlane_Alaska_2_FIPS_5002_Feet**
  
  o Then click OK to exit.

---

**5. Background Image**

To load the Lakina River aerial photo:

• **Select File | Add Data | Add Data or use the Add Data Shortcut tool 🌟. Select the Lakina_13 image in the “Workshop\DOT Images\Lakina River\Lakina_13 World” folder.**
  
  o ArcMap will automatically use the created World File.
  
  o Click Add.

• No, do not Pyramid build. Click No.

• An **Unknown Spatial Reference** massage will appear. Click OK to exit.

• **SAVE Project 📄.**

---

**6. Digital Elevation Data**

To load the elevation data:

**6.1. AutoCAD Survey Data**

• **Use the Add Data Shortcut tool 🌟. Attempt to add the LakinaRiver XML data in the “Workshop\DOT Scatter Data\XML” folder.**
  
  o GIS does not recognize XML formats, so let’s try the AutoCAD file.

• **Use the Add Data Shortcut tool 🌟. Select the file LakinaRiver.dwg in the “Workshop\DOT Scatter Data\AutoCAD” folder.**

• **An Unknown Spatial Reference** massage will appear. Click OK to exit. The AutoCAD data should appear and aligned with the Lakina aerial photo.

The elevation data need to be extracted as a shapefile:

• **In the TOC, click the “+” symbol next to “LakinaRiver.dwg Group Layer” to expand the layers.**
• R-click on the Point layer | Data | Export Data to open the Export Data dialog.
  
  o Make sure All features is selected in the Export drop-down menu and this layer’s source data is selected.
  
  o For the Output feature class: navigate to where you want to save the shapefile and name it as “LakinaRiverAutoCAD”.
  
  o Click OK.
  
• A message appear asking “Do you want to add the exported data to the map as a layer?,” select Yes.
  
• Now we can remove the AutoCAD Layer:
  
  o In the TOC, R-click “LakinaRiver.dwg Group Layer” | Remove
  
The Lakina River scatter data should be the only thing left.
  
• R-click on the LakinaRiverAutoCAD shapefile layer in TOC | Zoom to Layer.
  
This should reveal that extra points are present. To remove these extra points:
  
• Make sure the Editor Toolbars is activated.
  
• From the Editor drop-down menu, select Start Editing. Click Continue if a Spatial reference does not match data frame message appears.
  
  o R-click the LakinaRiverAutoCAD shapefile layer in TOC | Open Attribute Table.
  
  o R-click on Elevation column | Sort Ascending.
  
  o Select all rows with “0” Elevation values by holding Shift down. There should be roughly 547 rows with 0 elevation. The selected rows should turn blue.
  
  o R-click far left edge of selected rows | Delete Selected.
  
  o Close Attribute Table | Stop editing and Save
  
• From the Editor drop-down menu, select Stop Editing | Click Yes to save your edits.
  
• R-click on the LakinaRiverAutoCAD shapefile layer in TOC | Zoom to Layer.

Only the elevation data should now be present and aligned with the background image.
6.2. XML Survey Data

Unfortunately, the AutoCAD survey dataset does not include as much details (number of survey points) as the XML file that was used in SMS. And ArcMap do not see XML files.

We can export the XML data from SMS to a .txt file and then import it into GIS. To do this:

- Launch SMS from the desktop icon, start menu or from file browser.
- Select Open and navigate to “SMS Modeling Exercises\Exercise 1 – Background Data A” directory.
- Select the file named “Lakina River.sms.”, then select Open.
- Turn on the Lakina Scatter in the Project Explorer and click on the layer to activate it.
- File | Save As
  
  o Navigate to where you want to save the file.
  o For File name enter “Lakina River XML”.
  o In the Save as type drop-down menu, select “Tabular Data Files (*.txt).
  o Click Save. This should open the Export Tabular File dialog.
    
    ▪ Because the Lakina Scatter includes are x, y, & z data, the Number of Columns should have a 3 next to it.
    
    ▪ Change the Delimiter to Comma using the drop-down menu.
    
    ▪ Click on the Data button under Column 1 and select x location.
    
    ▪ Repeat for Column 2 and 3 using the following:
      
      • Column 1 = x location
      • Column 2 = y location
      • Column 3 = z

  o Click OK to exit.

- Close SMS.

In order to bring in the Lakina River XML.txt file into GIS we first need to add column headings. To do this:

- Use Excel to open the Lakina River XML.txt file | Comma delimiter. Change the column headings to x, y, z | save as an Excel Workbook.
The *Lakina River XML.xlsx* file can now be added into ArcMap.

- Back in ArcMap, select *File | Add Data | Add XY Data*.
  - In the *Choose a table...* drop-down menu find *Lakina River XML.xlsx*.
  - There should only be one sheet named ‘Lakina River XML$’, select it and click *Add*.
  - Make sure the *X Field* is “x”, *Y Field* is “y”, *Z Field* is “z”.
- Click OK.
  - A message saying *Table Does Not Have Object-ID Field* pops up, click OK to exit.

Notice the XML derived scatter set has more data points. Let’s make this layer a shapefile:

- R-click on the ‘Lakina River XML$’ Events layer | *Data | Export Data* to open the Export Data dialog.
  - Make sure *All features* is selected in the *Export* drop-down menu and *this layer’s source data* is selected.
  - For the *Output feature class*: navigate to where you want to save the shapefile and name it as “LakinaRiverXML”. Click *Save*.
  - Click OK.
- A message appear asking “Do you want to add the exported data to the map as a layer?,” select Yes.
- Now we can remove the ‘Lakina River XML$’ Events and LakinaRiverAutoCAD layers:
  - In the TOC, R-click on both layers | *Remove*

The Lakina River XML scatter data should be the only thing left.

- **SAVE Project**.

7. Adding Pier Locations into Terrain Data using ArcMap

Current version limitations of the 2D modeling capabilities in HEC-RAS: Cannot use the HEC-RAS bridge modeling capabilities inside of a 2D flow area.

An alternative option is to simply modify the terrain to include the bridge embankments, abutments, and even piers. This requires a little work in ArcMap by manually editing the terrain to include those features.
During the SMS modeling exercises, bridge pier locations were needed. We then cut out the mesh around the piers and the model treated the boundary conditions as a vertical wall. For HEC-RAS, we need the pier elevations, so the information can be added to the terrain.

- In Excel, navigate to Workshop\Bridge Data and open the LainaPiers file.
- For the pier elevations (z column), the height of the bridge deck was entered, which is 1389 feet.

Because we are adding in much higher elevation points into the middle of the channel, this will greatly affect the surrounding area when the TIN is developed. To eliminate a “cone” effect around the piers, ground elevation points need to be positioned around the base of the piers.

Three additional columns were added to the Excel file with pier surrounding “base” points. The “base” points were positions 1.5 feet from all of the 6 pier centers, because the piers are 3 feet in diameter, in the north, east, south and west directions. The “base” elevations were estimated from the nearest by survey points. To add in the “Pier” and “Base” elevation data:

- Close Excel.
- In ArcMap select File | Add Data | Add XY Data.
  - In the Choose a table... drop-down menu navigate to Workshop\Bridge Data and select LainaPiers$. Click Add.
    - Make sure the X Field is “x”, Y Field is “y”, Z Field is “z”.
  - Click OK.
- Repeat to add the base elevation points, but make sure the X Field is “xbase”, Y Field is “ybase”, Z Field is “zbase”. Click OK | OK
- Turn off the LakinaRiverXML layer in the TOC.
- R-click on the LakinaPeir$ Events layer in TOC | Zoom to Layer (Figure 1).

Figure 36: Zoomed in view of the bridge piers and base elevation points.
The 6 piers should be plotted and all with 4 surrounding “base” elevation points. To make the newly added layers shapefiles, do each of the following sets for both of the LakinaPier$ Events layers:

- R-click on the LakinaPier$ Events layer | Data | Export Data to open the Export Data dialog.

- Make sure All features is selected in the Export drop-down menu and this layer’s source data is selected.
  - For the Output feature class: navigate to where you want to save the shapefile and name the piers points “LakinaPier” and the base points “LakinePierBase”. Click Save.
  - Click OK.
  - A message appear asking “Do you want to add the exported data to the map as a layer?,” select Yes.
  - In the TOC, R-click on LakinaPier$ Events layer | Remove.

- SAVE Project.

8. Create TIN

To create in TIN using ArcMap:

- Click on the ArcToolbox | 3D Analyst Tools | Data Management | TIN | double click Create TIN to open the Create Tin dialog.

- In the Output TIN drop-down, navigate to where you want to save the file and name it “LakinaTIN”. Click Save.

  - Click OK

- Using the Input Feature Class drop-down menu, individually select LakinaPerBase, LakinaPier, and LakinaRiverXML (Figure 2). We are selecting all there layers because we want the TIN to be created using the points from the survey data, piers, and surrounding base points.
  - Click OK
• Wait a minute as the TIN is created. When the TIN is complete a little message will pop up in the lower right hand corner. The TIN should also automatically show in the map.

• In the TOC, check the elevation ranges for the LainaTIN. Make sure there are no 0 values.

![Create TIN dialog](image1)

Figure 37: Create TIN dialog.

![Lakina TIN](image2)

Figure 38: Lakina TIN.

• R-click on the LakinaTIN layer in TOC | ![Zoom to Layer](image3) (Figure 3).
As a result of the triangulation process, the downstream boundary has an artificial lip at the end, which acts as a wall. This bogus boundary will cause the prevention of modeled flow.

- To edit the TIN, the TIN editor needs to be activated:
  - Customize | Toolbars | TIN Editing.

- Using the TIN Editing drop-down menu, select Start Editing TIN (Figure 4).


  - Click on the Modify TIN Data Area tool. The Modify TIN Data Area dialog window appears.
    - Set the Selection to completely within polygon from the drop-down menu. This allows you to manually modify the TIN triangles that are completely within a digitized polygon.
    - Set the Mask to toggle current state from the drop-down menu. This changes the TIN triangle(s) to either on or off depending on the current state.

- By clicking on the map, digitize a polygon around the artificial lip at downstream boundary (Figure 5). Double-click to finish the digitized polygon. The triangles should immediately disappear.

- In the TIN Editing drop-down menu, select Stop Editing TIN | click Yes to save the edits to the Lakina TIN.

- SAVE Project.
9.1. **Breaklines**

Breaklines are also added using the TIN Editing tool. We will add breaklines to the whitish diagonal channels as we did in the SMS exercises Figure 6A.

- Toggle on the *LakinaRiverXML* layer so the survey points are visible and zoom into the whitish diagonal channels

- Using the *TIN Editing* drop-down menu, select *Start Editing TIN*. 
• Click on the Add TIN Line tool, which adds new breaklines to a TIN. The Add TIN Line dialog window appears.
  
  o Set the Line type to soft line from the drop-down menu.
    
    ▪ Line type—The type of breakline to be created, either hard or soft. Hard and soft qualifiers for line and polygon feature types are used to indicate whether a distinct break in slope occurs on the surface at their location. A hard line is a distinct break in slope, while a soft line will be represented on the surface as a more gradual change in slope.
  
  o Set the Height source to from surface from the drop-down menu.
    
    ▪ The elevation of the digitized line is interpolated from the selected locations on the surface.

• Double-click on the LakinaTIN layer in the TOC. Click the Display tab and inter 50% for the Transparency. The whitish diagonal channels should now be more visible.

• Draw in breaklines on either side of the whitish diagonal channels (Figure 6B). You do not have to click on a vertex like you do in SMS. The triangulations will automatically adjust as the breaklines are added.

• In the TIN Editing drop-down menu, select Stop Editing TIN | click Yes to save the edits to the Lakina TIN.

• SAVE Project.

10. Create Raster from TIN

You can convert a TIN to a raster to use in surface modeling. This conversion can be done using the Tin to Raster (3D Analyst) geoprocessing tool. The raster is created by interpolating its cell values from the elevation of the input TIN at the specified sampling distance.

• Click on the ArcToolbox | 3D Analyst Tools | Conversion | From TIN | double click TIN to Raster to open the TIN to Raster dialog.

• For the Input TIN, select LakinaTIN from the drop-down menu.

• In the Output Raster drop-down, navigate to where you want to save the file and name it “LakinaRaster”.

• Leave the Output Data Type as FLOAT.

• Leave the Method as LINEAR.
• Change the *Sampling Distance* to *CELLSIZE* using the drop-down menu.

• Click OK.

• Wait a minute as the Raster is created. When the Raster is complete a little message will pop up in the lower right hand corner. The TIN should also automatically show up in the map (Figure 7).

The raster now needs to be exported as a tiff file:

• Right click *lakinaraster* in TOC | *Data* | *Export Data* to open the *Export Raster Data* dialog.
  
  o For the *Location*, navigate to where you want to save it.
  
  o Make sure that *TIFF* is selected in the *Format* drop-down menu.
  
  o The *Name* should automatically populate as *lakinaraster1.tif*.

![Figure 42: Lakina River Raster, TIN and breaklines.](image)
• Click Save.

• An Output Raster message should appear asking “Would you like to add the exported data to the map as a Layer?”. Click Yes and the tiff file should plot.

- SAVE Project.

11. Conclusion

We have finished developing a terrain model in ArcMap. The terrain model is a requirement for 2D modeling, as it is used to establish the geometric and hydraulic properties of the 2D cells and cell faces. A terrain model is also need in order to perform any inundation mapping in HEC-RAS Mapper. The terrain model can now be imported into HEC-RAS.
Modeling Exercise #2 – Background Data

Gathering Data

1. Introduction

Currently HEC-RAS uses gridded data for terrain modeling. In the previous exercise a terrain model was created and exported so the gridded data can be read in by HEC-RAS.

Launch HEC-RAS 5.0.3 from the desktop icon, start menu or from file browser. HEC-RAS will open the “main window” menu bar (Figure 8).

Figure 43: HEC-RAS Main Window. Image from HEC-RAS_4.1_Users_Manual.
Save the project:

- Select File | Save Project As...
  - In the Save Project As dialog, navigate to where you want to save the file and enter a name of “LakinaRiver” and OK.
- In the HEC-RAS Main Window, the Project should have populated with LakinaRiver.

2. Setting the Spatial Reference Projection

The data specific spatial coordinate projection can be set using the RAS Mapper dialog.

- To open RAS Mapper, press the RAS Mapper button on the HEC-RAS Main Window.

To set the spatial reference system for the project:

- Select the Tools | Set Projection for Project menu item from the RAS Mapper menu bar. When the Set Projection option is selected a dialog window will appear.

To set the spatial reference system (coordinate system), we need to browse and select an existing “.prj” file (ESRI projection file) that contains the correct coordinate system. If ArcGIS is installed on the computer, the user can browse to the ArcGIS directory that contains a listing of all the available coordinate systems and select the appropriate one.

For this Lakina River example, the ArcGIS projection file (*.prj) has been saved in the Workshop\ESRI projection file folder.

- Navigate to the Workshop\ESRI projection file folder and select:
  - NAD_1983_2011_StatePlane_Alaska_2_FIPS_5002_Feet

3. Loading the Terrain Model

The next step is to load the terrain model that was created in Exercise #1.

- In RAS Mapper select the Tools | New Terrain...

At this time, RAS Mapper can import terrain data that is in the floating point grid format (*.flt); GeoTIFF (*.tif) format...

- Use the Plus (+) button to get a file chooser, then select the lakinaraster1.tif terrain layer from the Workshop\HEC RAS Modeling Exercises\Exercise 2 - Background Data folder. Click Open.
• Press the Create button to create the new Terrain Layer. Once the Create button is pressed, RAS Mapper will convert the grids into the GeoTIFF (*.tif) file format. Close the Creating Terrain dialog with the Terrain Complete appears.
  
  o Select Terrains to turn in on and R-click on Terrain | Zoom to layer (Figure 9).

• Save Project in the HEC-RAS Main Window.

Figure 44: Lakina terrain layer in RAS Mapper.

Once the terrain model is created the user can enhance the look of the terrain data by R-clicking on the terrain layer and selecting Layer Properties. The Layer Properties window (Figure 10) allows the user to: select and control the Surface Color Ramp; Transparency; Create and plot Contour Lines; and shade the terrain using a Hill Shading algorithm (Hill Shading makes the visualization of the terrain much more realistic and semi 3D).

Note: After a Terrain data set is created, the user will be able to display this terrain layer as a background image in the HEC-RAS geometry editor. Terrain layers, and any other Map Layers developed in RAS Mapper are available for display in the HEC-RAS Geometry editor.
4. **Background Image**

The next step is to load the Lakina River aerial:

- In RAS Mapper select the **Tools | Add Map Layer**
  - Change file type to “images” in the drop-down menu in the bottom right hand corner
  - Navigate to *Workshop\DOT Images\Lakina River\Lakina_13 World* and select *Lakina_13*. Click **Open**.
  - RAS Mapper will automatically use the created World File.
- R-click the *Lakina_13* image in the TOC | Image Display Properties
  - In the pop-up window, make the transparency ~50%. Click **OK**.

The background image and terrain layer should now both be visible.

- Close RAS Mapper.
- **Save Project** in the HEC-RAS Main Window.

5. **Conclusion**

The background data, including the ArcMap Terrain Model and aerial image have been read into HEC-RAS. It is now time to develop the 2D model.
Modeling Exercise #3 – Development of a 2D Model

Development of the 2D Computational Mesh

1. **Introduction**

   The HEC-RAS 2D modeling software capability uses a Finite-Volume solution scheme. This algorithm was developed to allow for the use of a structured or unstructured computational mesh. This means that the computational mesh can be a mixture of 3-sided, 4-sided, 5-sided, etc... computational cells (HEC-RAS has a maximum of 8 sides in a computational cell). However, the user will most likely select a nominal grid resolution to use (e.g. 200 x 200 ft cells), and the automated tools within HEC-RAS will build the computational mesh. After the initial mesh is built, the user can refine the grid with break lines and the mesh editing tools. A 2D computational mesh is developed in HEC-RAS by doing the following steps.

   To ensure consistency, read in a set of completed files from the end of exercise #2.

   - Open HEC-RAS and select Open
   - Navigate to “Workshop\HEC RAS Modeling Exercises\Exercise 2 - Background Data” directory
   - Select the file named “LakinaRiver”. It should populate the Title. Click OK.

2. **Drawing a Polygon Bounder for the 2D area**

   The user must add a 2D flow area polygon to represent the boundary of the 2D area using the 2D flow area drawing tool in the Geometric Data editor (just as the user would create a Storage Area).

   - Open the Geometric Data editor by click in the HEC-RAS Main Window.
   - Use the background mapping button on the HEC-RAS Geometry editor to turn on the terrain and other Map Layers if they existed, in order to visualize where the boundary of the 2D Flow Area should be drawn.
   - Select Lakina_13 Select Plot Terrain, click Close.
   - If the Terrain layer is not visible, you will need to go to the Geometry editor’s View menu, then select Set Schematic Plot Extents. From this window select the option called Set to Computed Extents. This option will reset the extents of the geometric
data editor view window to the extents of the terrain model you created and associated to the geometry data.

• *File | Save Geometry As*
  
  o In the *Save Geometry Data As* dialog, enter a *Title* of “LakinaGeo” and click OK.
  
  o In the HEC-RAS Main Window, the *Geometry: should have populated with LakinaGeo.*

To create the 2D flow area, use the 2D Flow Area tool located along the top of the Geometry editor.

• To draw the boundary of the 2D Flow Area, begin by left-clicking to drop a point along the 2D flow area polygon boundary. Then continue to use the left mouse button to drop points in the 2D flow area boundary. As the user runs out of screen real-estate, they can right-click to re-center the screen, this will give you more area to continue drawing the 2D flow area boundary. Double-click the left mouse button to finish creating the polygon (Figure 11).

![Figure 46: Example 2D flow area polygon.](image)

• Once the 2D area polygon is finished, the interface will ask the user for a Name to identify the 2D flow area. For this example enter: “2D Interior Area”.


Note: A 2D flow area must be drawn within the limits of the terrain model area being used for the study.

- R-click on the 2D flow area and select View Options to open the Geometry Plot Options.
  - Un-select Fill in Storage Area/2D Flow Areas. Close the options window.
- Save Project in the HEC-RAS Main Window.

### 3. Adding Break Lines inside of the 2D Flow Area

Before the computational mesh is created the user may want to add break lines to enforce the mesh generation tools to align the computational cell faces along the break lines. Break lines can also be added after the main computational mesh is formed, and the mesh can be regenerate just around that break line. In general, break lines should be added to any location that is a barrier to flow, or controls flow/direction.

![Figure 47: Example Breaklines.](image-url)
To add break lines by hand into a 2D flow area, select the 2D Area Break Line tool.

- Left click on the geometry window to start a breakline and to add additional points. Double click to end a break line. While drawing a breakline, you can right click to re-center the screen in order to have more area for drawing the breakline.

- Once a breakline is drawn the software will ask you to enter a name for the break line. Enter whatever name you want.

- Quickly add breaklines along the roads, and any river channel you want to align the mesh faces along. Breaklines can also be placed along the main channel banks in order to keep flow in the channel until it gets high enough to overtop any high ground berm along the main channel. An example of using breaklines within a 2D flow area is shown in Figure 12. Your breaklines do not have to match the image.

- **Save Project in the HEC-RAS Main Window.**

After all the break lines have been added, the computational mesh can be generated. Keep in mind the user can also add additional break lines after the mesh has been generated, and the computational mesh can be refined around an individual break line at any time.

### 4. Creating a Spatially Varied Manning’s Roughness Layer

Since we already created a Manning’s n coverage in the SMS, we are going to steal it and import it into HEC-RAS.

The SMS derived Lakina River Material shapefile can be found in the *Workshop\HEC RAS Modeling Exercises\Exercise 3 – Mesh* folder. The shapefile has already been exported from SMS for this exercise, but if you wanted to do it yourself, the steps are below:

- **In SMS | Click to highlight the Materials coverage in the Project Explorer | File | Save As.**
  - Change the *Save as type* to *Shape Files (*.shp)* in the drop-down menu.
  - Click Save
    - Select **Feature Polygons | Polygon Shapefile**

Open RAS mapper to read in the Lakina River Material shapefile

- **Select Tools | New Land Cover**
  - Click the “+” symbol and Navigate to the *Workshop\HEC RAS Modeling Exercises\Exercise 3 – Mesh* folder and select *LakinaLandUse_withN_values.shp | Open.*
• Verify the following Mann. N values are entered:
  • Channel: 0.035
  • Road: 0.015
  • Trees 0.1

  o Create!!!

• When the “Land Cover layer complete!” appears, close out of the dialog (Figure 13).

![Figure 48: Land Cover.](image)

• To changes to Land Cover colors R-clicking LandCover | Image Display Properties.

• Change to your desire.

Once the user has created a Land Cover layer in the *.tif file format, they need to associate that data layer with the geometry file(s) they want to use it with.

• To associate the Land Cover layer R-click on the Geometires (on the top left hand side of the RAS Mapper window) and select Manage Geometry Associations.
• Make sure for *Terrain* is has *Terrain*, and for *Land Cover* is has Landover.

Once a Land Cover layer is associated with a geometry file, the user can then build a table of Land Cover versus Manning’s n values, which can then be used in defining roughness values for 2D flow areas.

• Open the Geometric Data editor by clicking in the HEC-RAS Main Window.

• Use the background mapping button on the HEC-RAS Geometry editor to turn on the *Land Cover*. Select ☑ *Land Cover* then click Close.

• Click Tables | *Manning’s n by Land Cover* (very bottom).

• Because we imported the Manning’s n coverage in, the Default values are already set for us. Click OK to exit.

• Save Project in the HEC-RAS Main Window.

5. Creating the 2D computation Mesh

The HEC-RAS terminology for describing the computational mesh for 2D modeling begins with the 2D flow area. The 2D flow area defines the boundary for which 2D computations will occur. A computational mesh (or computational grid) is created within the 2D flow area. Each cell within the computational mesh has the following three properties (Figure 14):

![Figure 49: HEC-RAS 2D molding computation mesh terminology. Image from HEC-RAS 5.0 2D Modeling User’s Manual.](image-url)
• Cell Center: The computational center of the cell. This is where the water surface elevation is computed for the cell.

• Cell Faces: These are the cell boundary faces. Faces are generally straight lines, but they can also be multi-point lines, such as the outer boundary of the 2D flow area.

• Cell Face Points: The cell Face Points (FP) are the ends of the cell faces. The Face Point (FP) numbers for the outer boundary of the 2D flow area are used to hook the 2D flow area to a 1D elements and boundary conditions.

To create a 2D flow area computational mesh, select the 2D Flow Area editor button on the left panel of the Geometric Data editor (under the Editors set of buttons on the left) to bring up the 2D flow area editor window (Figure 15):

![2D Flow Area Editor](image)

Figure 50: 2D flow area Mesh Generator Editor.

The 2D Flow Area editor allows the user to select a nominal grid size for the initial generation of the 2D flow area computational mesh.

• To use this editor, first select the button labeled *Generate Computational points on regular Interval ....* This will open a popup window that will allow the user to enter a nominal cell size. The editor requires the user to enter a Computational Point Spacing in terms of DX and DY Enter the following:
  
  o Spacing DX = 20
  o Spacing DY = 20

• This defines the spacing between the computational grid-cell centers. Click *Generate Points in 2D Flow Area.*
Since the user can enter breaklines, the mesh generation tools will automatically try to “snap” the cell faces to the breaklines. The cells formed around breaklines may not always have cell faces that are aligned perfectly with the break lines. An additional option available is Enforce Selected Breaklines. The Enforce Selected Breaklines option will create cells that are aligned with the breaklines, which helps ensure that flow cannot go across that cells face until the water surface is higher than the terrain along that break line.

- Select Enforce Selected Breaklines (and internal Connections).
  - In the pop-up dialog Select All | OK to exit.

Default Manning’s n Value: This field is used to enter a default Manning’s n values that will be used for the cell outside of the Land Cover Classification to Manning’s

- Make sure the Default Manning’s n value is 0.06.
- Also click on the Edit Land Classification to Manning’s n to verify the correct classifications are entered. Click OK to exit.
- Leave Tolerances (Tol) as default values.

Now that the nominal grid size has been entered (20ft x 20 ft), breaklines have been selected, the base Manning's n-values has been verified and tolerances have been set, the mesh can be forced.

- Select Force Mesh Recomputation | OK.

When the OK button is selected the software automatically creates the computational mesh and displays it in the Geometric Data Editor graphics window (Figure 16).

Cells around the breaklines and the 2D flow area boundary will typically be irregular in shape, in order to conform to the user specified breaklines and boundary polygon. The mesh generation tools utilize the irregular boundary, as well as try to ensure that no cell is smaller in area than the nominal cell size.

- Save Project in the HEC-RAS Main Window.
The automated mesh generation tool in HEC-RAS works well, however, nothing is perfect. On occasion a bad cell will be created due to the combination of the user defined polygon boundary and the selected nominal cell size, or when the user is adding/modifying points inside of the polygon. After the mesh is made, the software automatically evaluates the mesh to find problem cells. If a problem cell is found, that cell’s center is highlighted in a red color (Figure 17), and a red message will show up on the lower left corner of the geometric data window.

6. Potential Mesh Generation Problems
Evaluate the Lakina River mesh to see if any problem cells (red cell centers) exist.

The HEC-RAS 2D Modeling User’s Manual covers many of the reasons for problem cells to occur and how to fix the problem. Adding points is a simple hand editing mesh manipulation tool that can fix most problems.

- Selects *Edit* then *Add Points*.
- Left-clicks anywhere within the 2D flow area and a new cell center will be added, and the neighboring cells are changed (once the mesh is updated) (Figure 18).
- Quickly add a few new points.

The entire mesh only updates once the user has turned off the editing feature, which saves computational time in creating the new mesh.

- To turn off the editing feature, Selects *Edit* then *Add Points*.
- Close the *Geometric Data Editor and Save Project* in the HEC-RAS Main Window.
7. **Running the 2D Geometric Preprocessor**

This is the option to pre-process the 2D flow area computational cells and faces into detailed tables based on the underlying terrain data. Running the 2D Geometric preprocessor occurs in RAS Mapper.

- To open RAS Mapper, press the **RAS Mapper** button on the HEC-RAS Main Window.

In the Geometry group there will be a sub layer called 2D flow area.

- Select 2D Flow to turn on the
- Right click on the 2D Flow sub layer, then select *Compute 2D flow areas Hydraulic Tables*.
- Right click on the 2D Flow sub layer, then select one of the tables (i.e. Cell Volume vs Elevation).
  - Take a minute to look at some of the tables.

8. **External 2D flow area Boundary Conditions**

There are five types of external boundary conditions that can be linked directly to the 2D flow areas. These boundary condition types are:

- Flow Hydrograph
- Stage Hydrograph
- Normal Depth
- Rating Curve
- Precipitation

The *Normal Depth* and *Rating Curve* boundary conditions can only be used at locations where flow will leave a 2D flow area. The flow and stage hydrograph boundary conditions can be used for putting flow into or taking flow out of a 2D flow area. For a Flow...
Hydrograph, positive flow values will send flow into a 2D flow area, and negative flow values will take flow out of a 2D area. For the Stage Hydrograph, stages higher than the ground/water surface in a 2D flow area will send flow in, and stages lower than the water surface in the 2D flow area will send flow out. If a cell is dry and the stage boundary condition is lower than the 2D flow area cell minimum elevation, then no flow will transfer. The Precipitation boundary condition can be applied directly to any 2D flow area as a time series of rainfall excesses.

For this example, a flow hydrograph boundary condition will be used to bring flow into the 2D area, while a normal depth boundary condition will be used for flow leaving the 2D area.

To add external boundary conditions to a 2D flow area:

- Open the Geometry Data editor by click \(\text{in the HEC-RAS Main Window}\)
- Select the tool (button) called SA/2D Area BC Lines.
- Draw a line along the outer boundaries of the 2D Area to establish the location of the boundary condition (Figure 19).
- Start with the inflow boundary line along the top. Double click to end the boundary condition line.
  - An interface will pop up asking to enter a name, enter “LakinaInflow” and click OK.
- A red and black line should appear.
- Repeat this process to add an outflow boundary line along the bottom. Name it “OutFlow”.


8.1. **Unsteady Flow Data editor**

Once all of the 2D flow area boundary conditions have been identified (drawn with the SA/2D Area BC Lines tool), the boundary condition type and the boundary condition data are entered within the *Unsteady Flow Data editor*. The *Unsteady Flow Data editor* is where the user selects the type of boundary condition and enters that boundary conditions data.

- Select *View/Edit Unsteady Flow Data* in HEC-RAS main window to open the dialog (Figure 20A).
Currently, none of the **Boundary Conditions Types** are activated. To activate them:

- Next to the **2D Interior Area BCLine: Outflow**, click in the blank space below **Boundary Condition**. Above, four **Boundary Conditions Types** should activate.

- For the Outflow **Boundary Conditions Type**, select **Normal Depth**. In the pop-up window, enter 0.01 for the **Friction Slope**. Click OK

- Storage/2D Flow Area of interest... **Boundary Condition**

- Next to the **2D Interior Area BCLine: LakinaInflow**, click in the blank space below **Boundary Condition**.

The inflow for a 2 year flood for the Lakina River is 1285 cfs. During the next sets, a single discharge value will be used to populate a Flow Hydrograph.

- For the Inflow **Boundary Conditions Type**, select Flow Hydrograph (Figure 20B).
  - In the Flow Hydrograph dialog, select **Enter Table**.
  - Change the **Data time interval** to 6 hours in the drop-down menu.
  - Select **Fixed Start Time**: and use the calendar to select 01JAN2016. Click OK.
  - Set the time to 0:00.
Enter 1285 for a 48 hours.

Enter a Min Flow: 0.

Enter 0.01 for the EG Slope for distributing flow along BC line.

OK to exit.

- To save, select File | Save Unsteady Flow Data. Enter “LakinaUnsteady” for the Title. Click OK.

- In HEC-RAS main window, the Unsteady Flow should be populated with LakinaUnsteady.

- Close the Unsteady Flow Data editor and save the project in the HEC-RAS Main Window.
Modeling Exercise #4 – Unsteady Flow Model

Unsteady Flow Simulation

1. Introduction

HEC-RAS has the ability to perform two-dimensional unsteady flow routing with either the Full Saint Venant equations (with added terms for turbulence modeling and Coriolis effects) or the Diffusion wave equations.

Within HEC-RAS the Diffusion Wave equations are set as the default and is what is used for this exercise.

To ensure consistency, read in a set of completed files from the end of exercise #3.

- Open HEC-RAS and select Open
- Navigate to “Workshop\HEC RAS Modeling Exercises\Exercise 3 - Mesh” directory
- Select the file named “LakinaRiver”. It should populate the Title. Click OK.

2. Performing the Computations

To run the model, open the Unsteady Flow Analysis window:

- Select Unsteady Flow Data in HEC-RAS main window to open the dialog.
- Save the Plan by selecting File | Save Plan As. Enter “LakinaPlan” for the Title and click OK.
  - Enter “LakinaPlan” for the short identifier and click OK.
- Under Programs to Run, Select Geometry Processor, Unsteady Flow Simulation, and Post Processor.

The Simulation Time MUST be the same as Start Time used in the Unsteady Flow Data editor.

- Using the calendar button, make the Starting Date: 01JAN2016 and enter a Time: 0:00.
- For the Ending Date and Time, that depends on how long you want the model to run. For the exercise set the Ending Date: 01JAN2016 and Time: 12:00.
• Set the *Computation Interval*: Between 30 and 1 seconds. For this exercise try 1 sec.

• Set the *Hydrograph Output Interval*: 5 min.

• *Mapping Output Interval*: 1 min.

• Click Compute!!! The HEC-RAS Computations window should appear (Figure 21).
  ○ Hopefully you get a **Finished Unsteady Flow Simulation**.

• *Save the project* in the HEC-RAS Main Window.

![HEC-RAS Computations window](image)

3. **Viewing Output using RAS Mapper**

Once the user has completed an unsteady-flow run of the model, the user can look at all of the 2D output results within RAS Mapper.

7.1 **Animating Map Layer**

Any Map Layer that is “Dynamic” can be animated in time. The animation control can be used to animate a single or multiple map layers.
To animate a single map layer, turn that map layer on, then make it the active map layer (Layer will be highlighted in a Magenta color). Once a layer is turned on, and made the active layer, then the animation control at the top of the map window, can be used to animate that layer in time. The animation control has a play button, as well as Max, and Min options.

- In RAS Mapper turn on the ☑ Results in the TOC and select ☑ Depth. Then activate the depth layer by clicking on the Depth.

- Turn off the ☐ Land Cover and the ☐ Geometries.

- Above the map window, in the upper right-hand corner click the “>” next to the green play Animation button ➔. Simulated water depth should emerge from the inflow boundary condition.
  - Click the “>” several times. Notice the time show below changes by 1:00 with every click (Figure 22).

Figure 57: RAS Mapper with the Depth results displayed.
• Click the Min button to bring the simulation back to the beginning.

• Click the green play button to watch a time lapse of the simulation. You can press the pause button at any time to stop the time lapse.

7.2 Plotting Velocity

RAS Mapper now has the ability to plot velocities spatially for 2D flow areas. Velocity is plotted with a color palette reflecting the magnitude of the velocity. Users can change the color palette, as well as the magnitude range for plotting the colors. Velocity vectors, which reflect direction and magnitude of the velocity, can be added to the plot. Additionally, there is an option to turn on a particle tracing visualization, which allows for much greater understanding of the velocity flow field, in both magnitude and direction.

• Turn on the Velocity output layer and turn Depth off. Then activate the velocity layer by clicking on it.

• Click Min then select the green play Animation button (Figure 23).

Figure 58: Color based velocity plot.
In addition to color velocity plotting, RAS Mapper has the option to add velocity vectors and show particle traces on top of the map layers.

- To add velocity vectors, press the `Static Velocity Arrows` button above the map window. This will turn on the velocity directions and magnitude arrows.

- To control the density of the arrows select the `Velocity Setting` button above the map window.

The `Velocity Map Parameters` settings window allows the user to control the spacing between arrows by selecting a Spacing (pixel width for the spacing between arrows). When the arrows are turned on, they are displayed in the direction of the velocity. The magnitude of the velocity is reflected in the size of the arrows (i.e. larger arrows equates to higher velocity).

- Take a minute to play around with the parameters.

Another extremely cool option for velocity plotting is the option called Particle Tracing. When this option is turned on, the user will see what appears to be particles of water moving through the flow field. This is a visualization of water particle movement to improve the understanding of the velocity and the direction of the flow.

- To turn this option on, press the `Particle Tracing` button.

Figure 59: Particle tracing visualization option.
Once this option is turned on, from the *Velocity Map Parameters* window the user can change the parameters that control the particle tracing visualization. These parameters are:

- **Speed** (Speed the particles move. The speed is a relative speed, it is not the actual speed of the particles).
- **Density** (density of the particles).
- **Width** (how thick they appear).
- **Lifetime** (how long a particle trace will last).
- **Anti-Aliasing** (Yes provides smoother lines for the particle traces, but takes more compute power.

4. Creating Static (Stored) Maps

The user can create a static map (map stored to the disk) at any time from RAS Mapper by selecting the **Tools** | **Manage Results Map** menu item. When this option is selected the window shown in (Figure 25) will appear.

![Results Mapping Window](image)

This editor will allow the user to create new map layers (Add New Map), as well as generate stored maps to a file (which can be used with HEC-FIA, or in a GIS).

- To create the stored map, first highlight the layer (i.e. **Depth**) to be created, then press the button labeled **Edit Map**.

- A **Results Map Parameters** window will appear (Figure 26).
  - Select the time of interest. For this example choose **Max**.
  - Under the **Stored (Saved to disk)** options, select **Raster based on Terrain**.
    - **Save Map**

- To create the stored map, highlight the layer **Depth** to be created, then press the button labeled **Compute/Update Stored Maps** in the upper right corner.
This will start the process of creating/updating stored maps for the stored map layers. When this process is complete, there will be a subdirectory within the project directory that is labeled the same name as the RAS Plan Short ID. This folder will contain the results in a gridded file format.

The user can now import the file into GIS to.

![Figure 61: Results Map Parameters.](image)

5. **Time Series Output Plots and Tables**

When Results Layer(s) are turned on for display, the user can also get time series plots and tables for those results layers.

- For example, if the velocity results map layer is turned on, right click on that map window over that layer and an option for *Time Series Plots* appears.

- Select *Velocity* and a plot will pop-up.
  
  - Because our model has a constant discharge of 1258 cfs, it will reach a steady state velocity and have a flat profile after an initial jump.

6. **Conclusion**

This workshop provided a basic introduction to setting up and running a HEC-RAS simulation. There are lots of ways to view the results, but they need to be saved and imported into another program to save them as an image.