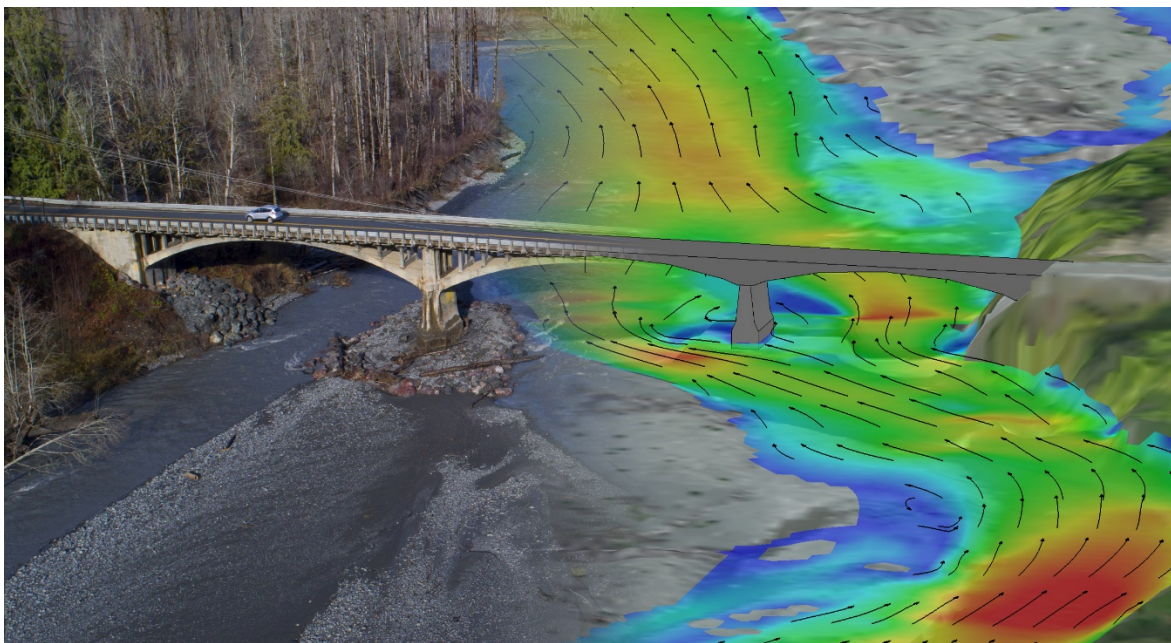# Two-Dimensional Hydraulic Modeling for Highways in the River Environment
Reference Document

U.S. Department of Transportation
Federal Highway Administration

Technical Report Documentation Page

1. Report No. FHWA-HIF-19-061	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle TWO-DIMENSIONAL HYDRAULIC MODELING FOR HIGHWAYS IN THE RIVER ENVIRONMENT – REFERENCE DOCUMENT		5. Report Date October 2019
		6. Performing Organization Code
7. Author(s) Dusty Robinson, Alan Zundel, Casey Kramer, Royd Nelson, Will deRosset, John Hunt, Scott Hogan, Yong Lai		8. Performing Organization Report No.
9. Performing Organization Name and Address Aquaveo, LLC Ayres Associates 3210 N. Canyon Road 3665 JFK Parkway Provo, Utah 84604 Building 2, Suite 100 Fort Collins, Colorado 80525 Northwest Hydraulic Consultants 711 Capital Way South, Suite 607 Olympia, Washington 98501		10. Work Unit No. (TRAIS)
		11. Contract or Grant No.
12. Sponsoring Agency Name and Address Federal Highway Administration Resource Center 300 8th Street, Room 826 Austin, Texas 78701		13. Type of Report and Period Covered
		14. Sponsoring Agency Code
15. Supplementary Notes Technical Assistance: Aquaveo – Todd Wood Ayres Associates - Matthew Hickox and Scott Zey Northwest Hydraulic Consultants – Achilleas Tsakaris, Matt George, and Katie Mozes		
16. This document is intended to be a comprehensive guide to the use of two-dimensional hydraulic modeling for highways in the river environment. With the help of this document, hydraulic engineers can become proficient in the application of 2D models for transportation projects. This document was funded through the Every Day Counts: CHANGE initiative		
17. Key Words Two-dimensional modeling, hydraulics, hydraulic modeling, open channel flow, hydraulic analysis	18. Distribution Statement	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 301	22. Price

TABLE OF CONTENTS

LIST OF FIGURES

vi

LIST OF TABLES

LIST OF SYMBOLS

a	=	acceleration, ft/s^2
α	=	velocity correction factor
A	=	cross section flow area, ft^2
C	=	Courant number
d	=	distance/depth above datum, ft
D_{50}	=	median grain diameter, median particle diameter, ft
$D_{xx, xy}$	=	dispersion terms
E	=	specific energy (depth plus kinetic energy head), ft
e	=	net mass input (rainfall + exfiltration – infiltration)
F	=	force, lb
Fr	=	Froude Number
g	=	acceleration due to gravity (constant), gravitational acceleration, ft/s^2
H	=	depth of water, ft
h_e	=	energy head loss between cross sections, ft
m	=	mass, slugs
M	=	momentum, slug-ft/s
n	=	time step index
n	=	Manning's roughness coefficient
Q	=	volumetric flow rate, discharge, ft^3/s
q	=	unit flow rate, ft^2/s
ρ	=	density, $slugs/ft^3$
t	=	time, time increment, s
$T_{xx, xy}$	=	depth-averaged turbulent shear stresses, lb/ft^2
U	=	depth-averaged velocity in the x-direction, ft/s
u	=	velocity of water at corresponding depth of water z, ft/s
V	=	cross section average velocity, or flow velocity ft/s
V	=	depth average velocity of water in the y-direction, ft/s
V_n	=	flow velocity at time step n, ft/s
V_{n+1}	=	flow velocity at time step n+1, ft/s
x, y	=	Cartesian coordinates

Y	=	cross section average depth of water, ft
y	=	hydraulic depth, ft
z	=	depth of water at corresponding velocity u, ft
z	=	WSE (bed elevation + H), ft
Z	=	elevation of channel bottom, ft
∂	=	partial derivative
Δ	=	rate of change
∇	=	del operator
$\int$	=	Integral symbol
dV/dt	=	rate of change of V at time t, or, derivative of V with respect to time
η	=	sediment fractional porosity
$\tau_{bx, by}$	=	bed shear stress, lb/ft^2
$\tau_{wx, wy}$	=	wind shear stress, lb/ft^2
Ω	=	Coriolis parameter
$\vec{q_s}$	=	unit sediment discharge

LIST OF ACRONYMS

3DEP	3D Elevation Program, USGS
AASHTO	American Association of State Highway and Transportation Officials
AREMA	American Railroad Engineering and Maintenance-of-Way Association
ASCII	American Standard Code for Information Interchange
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CHANGE	Collaborative Hydraulics: Advancing to the Next Generation of Engineering
csv	File type: comma-separated values
CVA	Consolidated Vertical Accuracy
DEM	File type: Digital Elevation Model
DIM	File type: Dense Image Matching
dsm	File type: digital surface model
dtm	File type: digital terrain model
ECW	File type: Enhanced Compression Wavelet (image)
EDC	Every Day Counts
ELJ	Engineered Log Jam
ERDAS	Earth Resources Data Analysis System
ESRI	Environmental Systems Research Institute
FEMA	Federal Emergency Management Agency
FESWMS	Finite Element Surface-Water Modeling System (2D model)
FHWA	Federal Highway Administration
FIS	Flood Insurance Study
flt	File type: float grid flle
FST2DH	Flow and Sediment Transport – Two-Dimensional Horizontal Plane (2D model)
FVA	Fundamental Vertical Accuracy
GCS	Global Coordinate Systems
GIS	Geographic Information System
GPS	Global Positioning System
HDS	Hydraulic Design Series
HEC	Hydraulic Engineering Circular
HEC-RAS	Hydrologic Engineering Center's River Analysis System
IFSAR	Interferometric Synthetic Aperture Radar
JALBTCX	Joint Airborne Lidar Bathymetry Technical Center of Expertise
JPEG	File type: commonly used method of lossy compression for digital images
LAS	File type: an industry-standard binary format for storing lidar data
Lidar	Light detection and ranging
LWM	Large Woody Material
MatLab	Desktop program that expresses matrix and array mathematics directly
MrSID	File type: image, multiresolution seamless image database
NAD 83	North American Datum of 1983

NAIP	National Agriculture Imagery Program
NAVD 88	North American Vertical Datum of 1988
NCHRP	National Cooperative Highway Research Program
NED	National Elevation Dataset
NFIP	National Flood Insurance Program
NGS	National Geodetic Survey
NGVD 29	National Geodetic Vertical Datum of 1929
NHI	National Highway Institute
NMAS	National Map Accuracy Standards
NOAA	National Oceanic and Atmospheric Administration
NRC	National Research Council
NSSDA	National Standard for Spatial Data Accuracy
NVA	Non-vegetated Vertical Accuracy
PC	Personal Computer
PNG	File Type: image, Portable Network Graphics
ppm	parts-per-million
RANS	Reynolds-average Navier-Stokes Equations
RMA2	Resource Management Associates – 2 (2D model)
RMSE	Root-Mean-Square Error
SMS	Surface Water Modeling System
SPZ	State Plane Zone
SRH-2D	Sedimentation and River Hydraulics – Two-Dimensional Model
SWE	Shallow Water Equations
TIFF	File type: Tagged Image File Format
UAS	Unmanned Aerial Systems
UAV	Unmanned Aerial Vehicle
UK	United Kingdom
US	United States
USACE	US Army Corps of Engineers
USDA	US Department of Agriculture
USGS	US Geological Survey
UTM	Universal Transverse Mercato
VVA	Vegetated Vertical Accuracy
WGS	World Geodetic System
WSE	Water Surface Elevation
WSPRO	Water Surface Profile

GLOSSARY

Term:	Definition:
abutment:	A structure built at the end of a bridge to support the lateral and vertical pressure of the arch or span.
alluvial:	Alluvial refers to sediment type created by running water. Particles often appear visibly rounded.
backwater:	Increase in water surface elevation relative to elevation occurring under natural channel and floodplain conditions. It is induced by a culvert, bridge, or other feature that obstructs or constricts the free flow of water in a channel.
bathymetry:	The below-water ground elevation.
base flood:	The flood or tide having a 1-percent chance of being exceeded in any given year.
Base Flood Elevation (BFE):	The BFE is the computed elevation to which flood waters are anticipated to rise during the base flood event.
boundary condition:	A location along the model domain where discharge and/or water surface are defined or set.
breaklines:	Lines generally drawn to define topographic features, such as retaining walls, curbs, tops of ridges, and streams.
bulk density (dry):	Density of the void and sediment mixture (mass per unit volume), including both the voids and sediment.
bulk density (submerged):	Density of the water sediment mixture (mass per unit volume), including both water and sediment.
calibration:	The process of adjusting model variables, constants, and parameters until the model output describes some observations.
cell:	A 2D cell is a 3, 4, or more-node shape; triangle, quadrilateral, or polygon – the collection of cells covers the domain of the model surface. (General term that can be used in place of "element" which is normally associated with specific models.)
Coriolis:	A mass moving in a rotating system experiences a force (the Coriolis force) acting perpendicular to the direction of motion and to the axis of rotation. On the earth, the effect tends to deflect moving objects to the right in the northern hemisphere and to the left in the southern and is important in the formation of cyclonic weather systems.
countermeasure:	An action intended to prevent, delay, or reduce the severity of hydraulic problems.

critical depth:	The depth of flow occurring when the inertial and gravitational forces are equal (i.e., Froude number equal to 1.0)
critical velocity index:	Ratio of stream velocity to velocity required to transport sediment. If greater than 1, then sediment would be expected to move.
culvert:	A tunnel, conduit, or pipe carrying a stream or open drain under a road or railroad (or other structure).
density (ρ)	A measure of the amount of mass per unit of volume, typically expressed as slug/ft^3.
deposition:	Deposition is the geologic process in which sediments, soil, and rocks are added to a landform or landmass.
Diffusion	The movement of particles from a region of higher concentration to a region of lower concentration.
diffusive wave:	Shallow water equation: The diffusive wave is valid when the inertial acceleration is much smaller than all other forms of acceleration, or in other words, when there is primarily subcritical flow.
discharge:	The amount of fluid passing a section of a stream - often in cubic feet per second.
discretize:	Represent a series or range using a discrete quantity range subset.
domain:	A specified area of model activity.
element:	A 2D element is a 3, 4, or more-node shape; triangle; quadrilateral; or polygon - the collection of elements covers the domain of the model surface.
empirical:	Based on, concerned with, or verifiable by observation or experience.
encroachment:	An action within the limits of the base flood plain.
fall-velocity	Settling velocity for sediment particles.
floodplain:	The land area susceptible to being inundated by floodwaters.
floodway:	Often Regulatory "Floodway." The floodplain area that is reserved in an open manner by Federal, State, or local requirements, i.e., unconfined or unobstructed either horizontally or vertically, to provide for the discharge of the base flood so that the cumulative increase in water surface elevation is no more than a designated amount.
flume:	A constructed channel for water in the form of an open sloped chute used for testing physical models.

fluvial system:	The natural river system consisting of (1) the drainage basin, watershed, or sediment source area; (2) tributary and mainstem river channels or sediment transfer zone; and (3) alluvial fans, valley fills and deltas, or the sediment deposition zone.
freeboard:	The vertical clearance of the lowest structural member of the bridge superstructure above the water surface elevation of the overtopping flood.
Froude number:	Dimensionless number that represents the ratio of inertial to gravitational forces in open channel flow.
gauge:	A site on a river, stream, canal, lake, reservoir, or other water body that measures or obtains systematic observations of water level or discharge.
gravity (acceleration of) (g):	The rate of acceleration resulting from gravitational forces, typically accepted as 9.81 meters per second squared or 32.174 feet per second squared, on earth.
grid:	A grid consists of a collection of nodes, connected to form cells (elements) that combine to form a piecewise surface.
hydraulics:	Applied science concerned with behavior and flow of liquids, especially in pipes, channels, structures, and the ground.
hydrodynamics:	The study of the motion of fluids and the forces acting on solid bodies immersed in fluids and in motion relative to them.
hydrograph:	The graph of stage or discharge against time.
hydrographic survey:	A survey of physical features present underwater.
hydrologic:	Science dealing with the occurrence, circulation, distribution, and properties of the waters of the earth and its atmosphere.
hydrology:	Science concerned with the occurrence, distribution, and circulation of water on the earth.
hydrostatic:	Pressure caused by water or fluids at rest.
incipient motion:	The hydraulic condition when bed material, often the median grain size, begins to move and sediment transport of bed material occurs.
interferometric:	Techniques in which electromagnetic waves are superimposed, causing the phenomenon of interference in order to extract information.
kinematic wave:	The kinematic wave equation is a simplification of the full SWE. It assumes that frictional forces equal gravitational forces and that other terms in the full SWE are negligible.

Lagrangian:	A Lagrangian model resolves forces acting on individual particles as they move through space over time.
lidar:	Aerial survey method that illuminates the target with pulsed laser light and measures the reflected pulses. Stands for Light Detection and Ranging.
low chord:	The low elevation of the lowest bridge span member.
meandering:	A meander is one of a series of regular sinuous curves, bends, loops, or turns in the channel of a river, stream, or other watercourse. Its sinuosity measures the degree of meandering of the channel. Streams or rivers with a single channel and sinuosities of 1.5 or more are defined as meandering.
median:	The 50th percentile point on a size distribution curve such that half of the units or particles (by weight, number, or volume) are larger and half are smaller (e.g., D_{50}).
mesh:	Mesh consists of nodes that are grouped to form a surface coverage of many three- or more-sided elements, cells, or zones.
model (hydraulic):	Hydraulic models, physical or numerical, are simplified representations of complex systems including flowing water, the channel bed and overbanks, vegetation, sediment and other transported materials, and the atmosphere.
momentum:	A quantity of motion of a moving body, measured as a product of its mass and velocity.
one-dimensional model:	One-dimensional model has variables (velocity, depth, etc.) changing predominantly in one defined direction, x, along the channel.
photogrammetry:	The science of making measurements from photographs, especially for recovering the exact positions of surface points.
planimetrics:	The study of plane measurements, including angles, distances, and areas. Planimetric elements in geography are those features that are independent of elevation, such as roads, building footprints, and rivers and lakes.
porosity	The percentage of volume not occupied by solid particles inside a given volume of sediment, obtained by dividing the volume of voids by the total volume, or one minus the bulk density divided by the particle density.
rating curve:	A rating curve is a graph of discharge versus stage for a given point.
reach:	A length of a stream or river with beginning and end points selected arbitrarily or for geographic or historical significance.

regime:	The general pattern of variation around a mean condition, as in; flow regime, tidal regime, channel regime, sediment regime, etc. (also used to mean a set of physical characteristics of a river).
revetment:	Rigid or flexible armor placed to inhibit scour and lateral erosion.
rigid bed model:	Hydraulic model where the surface is fixed - no sediment transport is modeled.
riparian:	Relating to or situated on the banks of a river.
riprap:	Rock or other material used to armor shorelines, streambeds, bridge abutments, pilings, and other shoreline structures against scour and water or ice erosion.
roughness coefficient:	Numerical measure of the frictional resistance to flow in a channel, as in the Manning's or Chezy's formulas.
shear stress:	Stress or drag developed at the channel bed by flowing water. For uniform flow, this force is equal to a component of the gravity force acting in a direction parallel to the channel bed on a unit wetted area. Typically expressed as lb/ft^2.
Shields parameter:	Is a non-dimensional number used to calculate the initiation of motion of sediment in a fluid flow.
simulation:	A discrete computer run of a hydraulic model to generate results for the purpose of study.
sinuous:	Having many curves, bends, and turns. The sinuosity of a watercourse is the ratio of the length of the channel to the straight line down-valley distance.
skewed:	An oblique angle; a slant.
specific gravity:	The ratio of the density of a material to the density of water, typically taken as the dry bulk density divided by the density of water.
stratigraphy:	How rocks are layered. Geology concerned with the study of rock layers, primarily seen with sedimentary and layered volcanic rocks.
stream power:	Stream power is the rate of energy dissipation against the bed and banks of a river or stream per unit downstream length.
streamwise:	In the direction of the stream flow - conceptually along main general flow line.
subcritical flow	Subcritical flow occurs when the actual water depth is greater than critical depth. Subcritical flow behaves in a slow or stable way. It is defined as having a Froude number less than one.

supercritical flow:	Supercritical flow occurs when the actual water depth is less than critical depth. Supercritical flow behaves as rapid and unstable. It is defined as having a Froude number greater than one.
surcharge:	The additional water storage volume or elevation between the existing and the design water level.
tidal:	Relating to or affected by tides - swings in water surface elevation due to celestial bodies.
topographic:	Relating to the arrangement of the physical features of an area.
transverse:	Across, at right angles to, the main direction of flow.
tributary:	A stream that flows into a larger stream, river, or lake.
turbulence:	Motion of fluids in which local velocities and pressures fluctuate irregularly in a random manner as opposed to laminar flow where all particles of the fluid move in distinct and separate lines.
two-dimensional model:	A numerical hydraulic model that computes two components of velocity, usually the horizontal velocity components.
Unmanned Aerial System (UAS):	An unmanned aircraft that can fly autonomously, under remote control, or under direct human control. May also be referred to as an unmanned aerial vehicle (UAV) or drone.
uniform flow:	Flow of constant cross section and velocity through a reach of channel at a given time. Assumes both the energy slope and the water slope are equal to the bed slope under conditions of uniform flow.
unsteady flow:	Flow of variable discharge and velocity through a cross section with respect to time.
velocity:	Velocity is the speed of an object in a given direction. Velocity is a vector quantity, both direction and magnitude are important
verification:	The process of determining how well a calibrated model agrees with some observations.
viscosity:	The viscosity of a fluid is the measure of its resistance to gradual deformation by shear stress or tensile stress.
viscosity (dynamic) (μ):	A measure of internal fluid friction resisting flow in units of force per unit area. 2.05×10^{-5} lbs/ft^2 in freshwater at 70°F.
viscosity (kinematic) (v)	The ratio of dynamic viscosity to density, a quantity in which no force is involved. 1.06×10^{-5} ft^2/ s in freshwater at 70°F.
viscous fluids:	Viscous fluids have resistance to gradual deformation by shear stress or tensile stress.

watershed:	The upstream area of land that drains to a specific location - i.e., the land upstream of a bridge specifically draining to the bridge.
waterway:	Generally, a river, canal, or other body of water serving as a route or way of travel or transport.
weir:	Low dam, designed to overflow, often across a channel, and used to control stage or long term channel erosion.
wetland:	Areas where water covers the soil or is present either at or near the surface of the soil all year or for varying periods during the year, including during the growing season.
willow:	A shrub or tree of temperate climates that typically has narrow leaves, bears catkins, and grows near water.
zonal classification:	Classification based data set value (ranges) and/or material properties or other attributes specified for polygonal regions as is common with GIS products.

ACKNOWLEDGMENTS

NOTICE

QUALITY ASSURANCE STATEMENT

NO BINDING EFFECT

CHAPTER 1

INTRODUCTION

This reference document is intended to be a comprehensive guide to the use of two-dimensional (2D) hydraulic modeling for highways in the river environment. It emphasizes applications that are important to surface transportation engineering. This document focuses primarily on riverine hydraulic analyses, which may include the use of tidal boundary conditions. The content does not apply to coastal hydraulic modeling. For more information related to coastal modeling refer to the Federal Highway Administrations (FHWA) "A Primer on Modeling in the Coastal Environment" (FHWA 2017).

Application of 2D hydraulic modeling (2D modeling) is rapidly becoming the preferred analytical approach across a broad spectrum of study types, including transportation hydraulic studies, bridge scour, flood control design, river restoration, habitat evaluation, floodplain mapping, and many others. The interface of rivers and highways presents many design challenges that favor a 2D modeling approach. While many modeling practices and approaches are common to all types of studies, some concerns are specific to highway facilities, such as accurate modeling of bridge waterways, roadway overtopping, longitudinal roadway encroachments, and determining hydraulic variables for bridge scour calculations. The purpose of this document is to assist hydraulic engineers in becoming proficient in the application of 2D models to transportation projects.

Unless otherwise indicated (by context or citation to statute or regulations), the "requirements" discussed in this document are principles or best practices necessary to ensure scientific reliability, modeling accuracy, or the fidelity of modeling results. This document does not create any new legal requirements.

1.1. BACKGROUND

Since 2016, the use of 2D modeling for transportation projects has dramatically increased as 2D modeling has become more feasible for transportation agency staff and their consultants. This increase is attributable to several factors, including:

- Improved computational power of personal computers, making it feasible to run computationally intensive models.

- Significantly increased availability of higher quality digital ground elevation data, such as light detection and ranging (lidar), in the public domain.

- Improved numerical stability and relative ease of use of modern 2D modeling programs.

- Introduction of time-saving model development tools in graphical user interfaces.

With encouragement and support from FHWA, many transportation agencies and their consultants are now applying 2D modeling to a growing number of projects. FHWA promotes the use of 2D modeling through several resources, including Hydraulic Engineering Circular (HEC) No. 18 "Evaluating Scour at Bridges, Fifth Edition" (FHWA 2012) and Hydraulic Design Series (HDS-7) "Hydraulic Design of Safe Bridges" (FHWA 2012b). As stated in HDS-7, FHWA strongly recommends 2D modeling over one-dimensional (1D) modeling for design scenarios involving

complex flow conditions. FHWA has developed training through The National Highway Institute (NHI) based on these manuals, including NHI course #135046, "Stream Stability and Scour at Highway Bridges" and course #135090, "Hydraulic Design for Safe Bridges."

The FHWA provides training for 2D modeling via the NHI course #135095, "Two-Dimensional Modeling of Rivers at Highway Encroachments." The promotion of the technology and availability of training is another reason for the increase in 2D modeling in recent years. In 2017 FHWA included promotion, technology transfer, and training for 2D modeling in the initiative known as Collaborative Hydraulics: Advancing to the Next Generation of Engineering (CHANGE), which is part of the Every Day Counts program (EDC: CHANGE). Through this program, FHWA continues to advance the state of practice in transportation hydraulics to include 2D modeling in many state transportation agencies. Informational, technical support, and training resources are being developed and provided to states to assist in the development of their 2D modeling capabilities.

This reference document is the primary source document produced under the EDC CHANGE initiative. It provides engineers, designers, and other professionals involved in delivering transportation projects with extensive insight into the framework, data requirements, development process, and use of 2D models.

1.2. HISTORY OF TRANSPORTATION HYDRAULIC MODELING

Since the mid-1800s, hydraulic engineers in the United States (US) have sought reliable methods of dealing with the interface of rivers, floodplains, and transportation facilities (Figure 1.1). The earliest methods of sizing hydraulic openings were based on experience and historical performance. However, as the railroads expanded westward, there was no flood history or other upstream or downstream structures to use as the basis for determining bridge or culvert size. This led to the development of tabular and empirical methods that related waterway opening to factors that accounted for drainage basin and stream characteristics. The American Railroad Engineering and Maintenance-of-Way Association (AREMA) published a report in 1911 that presented formulas for waterway area and design discharge (AREMA 1911).
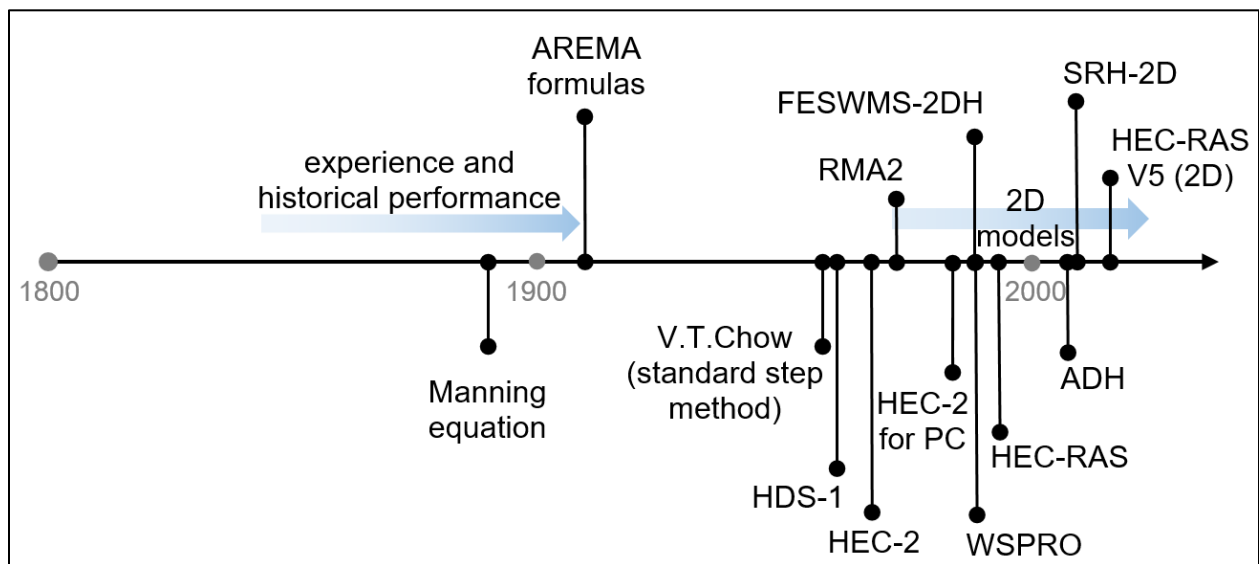
Figure 1.1. Timeline of transportation hydraulic analysis in the US (only publicly available models shown).

In the same timeframe in Ireland, Robert Manning was developing what would come to be known as the Manning Equation (Manning 1891). The Manning Equation, based upon several hydraulic formulas of the time, establishes a relationship between flow velocity, the size and shape of the channel, the channel slope, and the resistance to flow for steady and uniform flow conditions. Despite being an improvement to hydraulic analysis, the Manning Equation did not consider backwater, other non-uniform flow effects, and unsteady flow conditions.

Over half a century later "Open-Channel Hydraulics" (V.T. Chow 1959) presented the standard-step method, a numerical technique for calculating a water surface profile in a natural channel or floodplain by using the principles of conservation of mass and conservation of energy. To account for non-uniform flow conditions, FHWA published HDS-1 "Hydraulic Design of Bridge Waterways" in 1960 (updated in 1970 and 1978 (FHWA 1978)), which presented equations and nomographs (i.e., graphical calculating devices) to estimate the backwater caused by a highway crossing based on a specific bridge geometry and flow angle.

Based on Chow's work, engineers developed computer software for computing water surface profiles with the standard-step method. In the late 1960s, the US Army Corps of Engineers (USACE) Hydrologic Engineering Center (HEC) released the program "Backwater Any Cross Section" that would come to be known as "HEC-2 Water Surface Profiles" (USACE 1991). HEC-2 became the most popular program in the USACE HEC series. This program continually evolved and, in 1984, was made available in the personal computer (PC) environment.

HEC-2, a 1D model, marks the transition of bridge hydraulic analysis in the US from basic mathematical equations to numerical models. In this context, 1D means that the variability of hydraulic components such as flow depth and velocities are considered only in the streamwise direction (upstream or downstream) in the governing equations. Any components of hydraulic forces that do not act in the streamwise direction are neglected. Numerical models come in a variety of types, including 1D, 2D, and 3D, that provide a range of approximations of basic hydraulic principals (Figure 1.2). Numerical models have been the primary method of bridge hydraulics after the introduction of HEC-2.

| 1D model geometry | 2D model geometry | 3D model geometry |
| 1D model output | 2D model output | 3D model output |
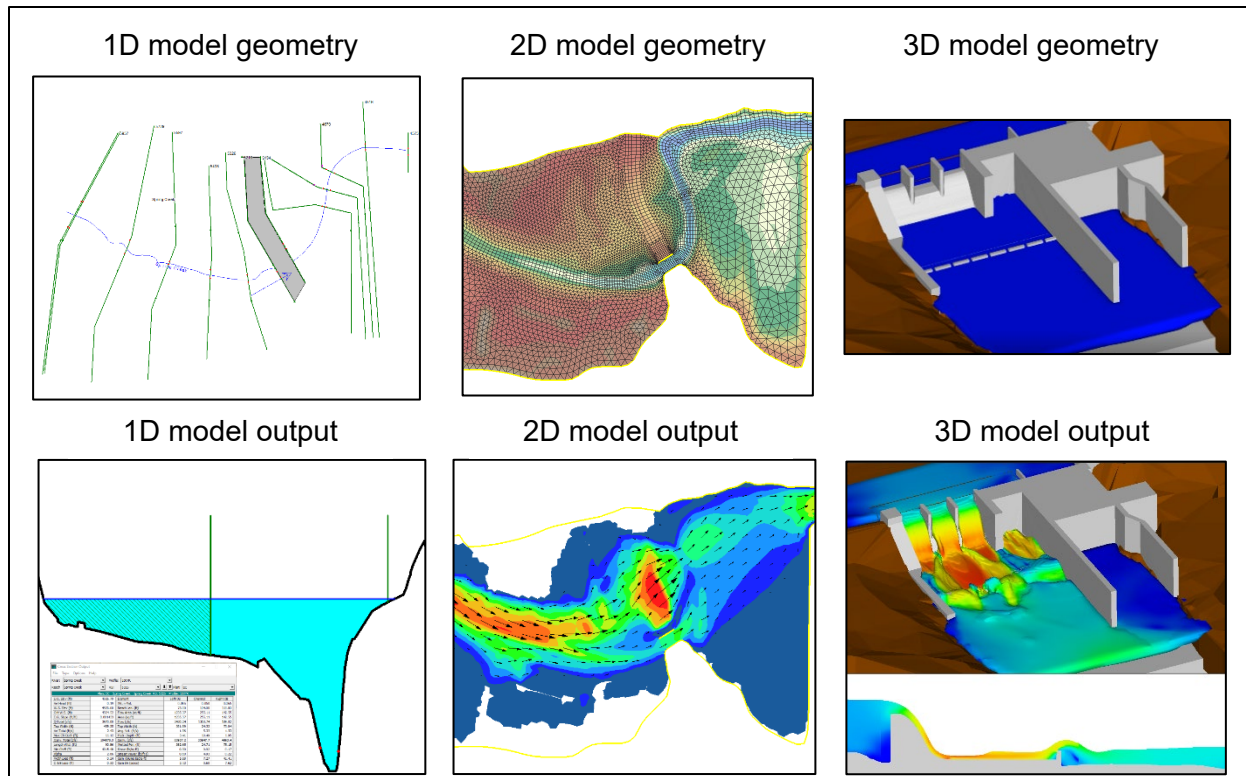
Figure 1.2. Example 1D, 2D, and 3D model geometry and output.

FHWA, in cooperation with the United States Geological Survey (USGS), also developed a computer program for water surface profile computations, but with particular emphasis on bridge waterways. The USGS/FHWA water surface profile computation model would become known as WSPRO (FHWA 1986). Both the 1D HEC-2 and WSPRO models were developed to compute water surface profiles for rivers and floodplains, with natural cross sections, using the standard-step method.

In 1995, the USACE made the first release of the HEC's River Analysis System (HEC-RAS). This program replaced HEC-2 with improved computations and, more significantly, with a graphical user interface for developing input and viewing results. HEC-RAS incorporated aspects of the WSPRO methodology as a bridge modeling option for users. HEC-RAS quickly became the 1D modeling method of choice for hydraulic engineering studies throughout the US, including those carried out for transportation agencies. For roughly two decades, HEC-RAS's capabilities were strictly limited to 1D modeling. The release of Version 5 in 2016 added 2D modeling functionality, in addition to the USACE's other 2D model the Adaptive Hydraulics Model System (ADH).

Up until the 1990s, 2D models could only run on powerful mainframe computers because of their computational intensity. Processors for PCs became powerful enough in the early 1990s to allow mainstream use of 2D modeling. Since that time, the number of projects using 2D modeling for transportation applications has steadily increased. This is in part because of the long-recognized limitations and inaccuracies of 1D models for complex hydraulic situations, such as those often encountered at bridge waterways. Several computer programs are now available to perform 2D modeling in addition to the previously mentioned HEC-RAS Version 5 (see Section 2.4).

Early 2D models prevalent in the US included the USACE's RMA2 and the USGS's FESWMS-2DH (Froehlich, 1989), the first 2D model endorsed by FHWA. Its application to transportation

projects began in 1988. FHWA sponsored, and the University of Kentucky conducted the development of FESWMS-2DH as a support modeling program. In 2003 the FESWMS-2DH program name was changed to Flow and Sediment Transport Model (FST2DH), and FHWA published a User's Manual (FHWA 2003).

In 2012, FHWA published HDS-7. This hydraulic design series document, which is not legally binding, established several best practices in hydraulic modeling for bridge design. While HDS-7 considers and explains both 1D and 2D modeling, it strongly encourages the use of 2D modeling for certain situations.

In 2013 FHWA began using the Sedimentation and River Hydraulics model (SRH-2D), developed and maintained by the US Bureau of Reclamation (Reclamation), and recommended its use for transportation hydraulic analyses. SRH-2D marks a shift from the use of FST2DH. SRH-2D avoids the stability issues common with earlier 2D models like RMA2 and FST2DH. The Surface Water Modeling System (SMS) software added SRH-2D added as a supported modeling program in 2013. FHWA partnered with Reclamation to incorporate several advanced hydraulic structure modeling features into SRH-2D and has supported the development of a custom graphical user interface in SMS.

1.3.   SCOPE OF THE DOCUMENT

With powerful software tools available to modelers, it is possible, though not recommended, to develop and run 1D and 2D models with little background in river hydraulics and related disciplines. However, to ensure that important aspects of hydraulic modeling are not overlooked, resulting in inaccurate model results, it is imperative that modeling practitioners understand the underlying subject matter. Useful 1D models necessitate more background in hydraulics to properly develop and execute, compared to 2D models, due to significantly more assumptions required (See Section 2.3). One primary advantage of 2D models is the reduction in model uncertainty because 2D models calculate many of the parameters assumed when using 1D models. Creating useful 1D models of project sites with complex hydraulics can be very difficult, if not impossible, even with sufficient background and experience in hydraulics.

While developing 2D models requires fewer assumptions than 1D models, it is important to have skill sets such as a background in, and an understanding of, open-channel flow, river and floodplain hydraulics, characteristics of hydraulic structures and controls, and variations in resistance to flow within channels and floodplains. In addition to having a background in general hydraulic engineering, modelers need training, knowledge, and skills specific to developing and using 2D models including evaluating and using geospatial data.

The advancement and proliferation of the tools, data, and software for 2D modeling in recent years has often outpaced the necessary documentation and training to support hydraulic engineers in applying 2D modeling to projects. This document assumes that the user has the requisite background in basic hydraulics and builds on this background to convey the specific knowledge and skills necessary for 2D modeling.

The goal of this document is to provide an engineer with the tools and knowledge to plan, develop and run a 2D model, and examine and post-process the results in support of project design. The material provided herein is intended to be useful for a broad range of available 2D modeling software. To the extent possible, the document avoids reference to one specific software product to the exclusion of another. However, Appendix A provides extensive instruction on the use of SRH-2D within SMS since this is the 2D surface-water model currently recommended by FHWA

for bridge hydraulic design and scour analyses. This document currently does not provide detailed program-specific information for any other 2D modeling software.

1.4.   ORGANIZATION OF THE DOCUMENT

The organization of the document is as follows:

Chapter 2. 2D Modeling Overview

This chapter describes the assumptions and advantages of 2D modeling over 1D modeling, as well as limitations of both 1D and 2D models. Information is provided on 2D modeling software currently available and on selecting the appropriate software for a given project type or situation. It also provides information on the future of 2D modeling and three-dimensional (3D) computational fluid dynamics.

Chapter 3. 2D Modeling Fundamentals

This chapter explains the underlying equations and principles, simplifying assumptions, and numerical solution approaches for the range of available 2D modeling tools. It also explains the different types of user-entered boundary conditions and how they influence hydraulic modeling.

Chapter 4. Data and Modeling Requirements

This chapter describes the data to be acquired, processed, and developed to perform a 2D modeling project. It includes discussions of data types and methods of finding, collecting, evaluating, and using terrain data; assigning boundary and initial conditions; determining and assigning roughness values; and data required for sediment transport studies.

Chapter 5. 2D Model Development Process
This chapter explains the workflow and planning of 2D models, determining model domain extents, developing geometric meshes, accommodating hydraulic controls and hydraulic structures, defining boundary conditions, entering roughness values, and running models.

Chapter 6. Model Review and Calibration

This chapter covers methods to evaluate model results, along with the data requirements and process for calibrating and verifying a model. It also provides tips for model documentation.

Chapter 7. Presentation of Model Results

Software applications available for 2D modeling provide many useful and visually informative ways to present results. This chapter provides tips for viewing the results in plan view and extracting the results in profile and cross section form. It also explains how to automate additional calculations within the graphical user interface for geospatial analysis.

Chapter 8. Modeling Application

This chapter provides 2D modeling strategies for various scenarios relevant to transportation engineering. Scenarios covered include road crossings of rivers, floodplain mapping and compliance studies, river rehabilitation and aquatic habitat analysis, and sediment transport.

Chapter 9. References

This chapter lists the literature cited, image references, and hyperlinks in the document.

Appendices

The appendices of this document contain information on the following topics:

Appendix A: Using SRH-2D with the SMS Graphical User Interface
Appendix B: Example Survey Request Form
Appendix C: 2D Model Review
Appendix D: 2D Model Report Outline

1.5. KEY TAKEAWAYS

The following list includes key takeaways that modelers should understand and consider before developing 2D models. Referenced chapters include further discussion on each point.

- 2D models eliminate the numerous assumptions required by 1D models (chapter 2);

- Acquiring accurate terrain data and developing a mesh that represents it is one of the most important steps to developing useable 2D models (chapter 4 and 5);

- 2D models do not necessarily need more terrain data than 1D models, though more complete data is better, but can provide betters results given the same terrain data for both models (chapter 4);

- Modelers should be able to identify primary hydraulic controls and sufficiently represent those controls in the mesh (chapter 4 and 5);

- Before starting a new model, the modeler should be able to answer the question, "What is this model for?", which directs the entire model development process (chapter 5);

- Mesh element faces should align with hydraulic controls (chapter 5);

- Modelers should seek to develop an efficient mesh/model (chapter 5);

- Calibrating a model to known data, or sensitivity testing in the absence of known data, is an important step in the process of developing a useable hydraulic model (chapter 6).

CHAPTER 2

2D MODELING OVERVIEW

Hydraulic models are simplified representations of complex systems including flowing water, the channel bed and overbanks, vegetation, sediment and other transported materials, and the atmosphere. As John von Neumann stated, "The truth is much too complicated to allow anything but approximations" (Neumann 1947). Models are developed to gain insight into these processes, either to understand the nature of these interactions (in which case we are performing analysis) or to guide modifications of these systems in beneficial ways (performing design). Numerical hydraulic models are simplified mathematical representations of this reality.

2.1. MODEL UNCERTAINTY

Models include varying levels of uncertainty because they are simplifications of reality. It is the responsibility of the modeler to understand the types of uncertainty because they set the limitation of the application of the model. Models have three types of uncertainty, as explained below, in order of decreasing magnitude.

2.1.1. Input Uncertainty

Models are only as good as the underlying data from which they are constructed. It is possible to build a high-resolution model from sparse or flawed data, but the accuracy of the underlying data limits the usefulness of hydraulic results. Modelers should exercise judgment regarding the accuracy of any input data and the resulting hydraulic model. Understanding the quality and limitations of the data the model uses, guides modelers in the correct application of the results. Chapter 4 provides more information regarding modeling data requirements.

2.1.2. Uncertainty of Physical Assumptions

The laws of physics govern how water interacts with its surroundings. Currently, it is not practical to numerically simulate every molecule of water interacting with every other molecule of water, air, or sediment. Therefore, all hydraulic models incorporate numerous physical assumptions and simplifications. For example, many hydraulic analyses assume a rigid bed (See Section 3.6.2).

Physical processes such as the Coriolis effect, cavitation, wind shear, and many others are often neglected in numerical models but can have impacts in the real world. Modelers need to be aware of what physical processes are included or ignored by the model they are using. The following sections provide more information about the general assumptions and limitations made by the principal types of models available. Chapter 3 includes additional sections related to underlying equations, assumptions, approaches, and simplifications.

2.1.3. Numerical Uncertainty

Numerical models routinely perform millions of computations which necessitate simplifying approaches for efficient model execution. Common approaches to simplify models result in some level of numerical uncertainty. Small differences in numerical methods can produce variances in results. The same input data and parameters used in two different models, or versions of the same model, would provide different results. See Chapter 6 for more information about model calibration and verification.

The impact of each category of uncertainty varies depending on the type of model used. For example, 1D models make significantly more simplifying assumptions compared to 2D models, and similarly, 2D models make more assumptions than 3D models. For the last four decades, 1D modeling has been the industry standard for a broad range of hydraulic analysis and is most likely the type of hydraulic model most modelers have experience using. Because of this legacy, and for comparison purposes, the following sections present the advantages and limitations of both 1D and 2D models.

2.2. ADVANTAGES, ASSUMPTIONS, AND LIMITATIONS OF 1D MODELING

One-dimensional models are extremely useful tools that simplify the 3D world into a 1D framework. One-dimensional models represent a waterway with cross sections and associated attributes specified along and between each section. Input data and computed model results are associated with these user-defined cross sections.

While 2D models are becoming the primary tool for hydraulic analysis, 1D models are still useful when the hydraulics are relatively simple and do not violate the internal assumptions of 1D models and when modelers can confidently make the assumptions required for input. For example, flow in a canal is relatively simple and can be analyzed successfully with a 1D model. However, a river with a wide floodplain and skewed roadway with multiple bridge openings is too complex for a 1D model. Chapter 4 of HDS-7 provides additional guidance for bridge hydraulic model selection. HDS-7 recommends 2D models for all transportation infrastructure projects involving complex hydraulics. The recommendation becomes more significant for scenarios that include any of the following conditions:

- Wide floodplains
- Highly variable floodplain roughness
- Highly sinuous channels
- Multiple embankment openings
- Moderate to high roadway skew

- Multiple channels
- Large tidal waterways
- Upstream hydraulic controls
- Roadway overtopping

2.2.1. Governing Equations

The first step in understanding the advantages and limitations of a numerical model is to understand how they produce hydraulic results. HEC-RAS 1D uses continuity and the standard step solution of the energy equation (Figure 2.1) to produce a water surface profile for steady (constant discharge) or unsteady (variable discharge) conditions.

The governing equations for 1D modeling shown below do not account for the many nuances of flow such as flow direction or path, velocity distribution, momentum, turbulence, or diffusion, and involve the following assumptions and limitations:

$$Q = VA \tag{2.1}$$

where:

Q = volumetric flow rate
V = cross section average velocity
A = cross section flow area

$$Z_2 + Y_2 + \frac{\alpha_2 V_2^2}{2g} = Z_1 + Y_1 + \frac{\alpha_1 V_1^2}{2g} + h_e \tag{2.2}$$

where:

1	=	denotes downstream section
2	=	denotes upstream section
Z	=	elevation of channel bottom
Y	=	cross section average depth of water
V	=	cross section average velocity magnitude
α	=	velocity correction factor
g	=	gravitational acceleration
h_e	=	energy head loss between cross sections
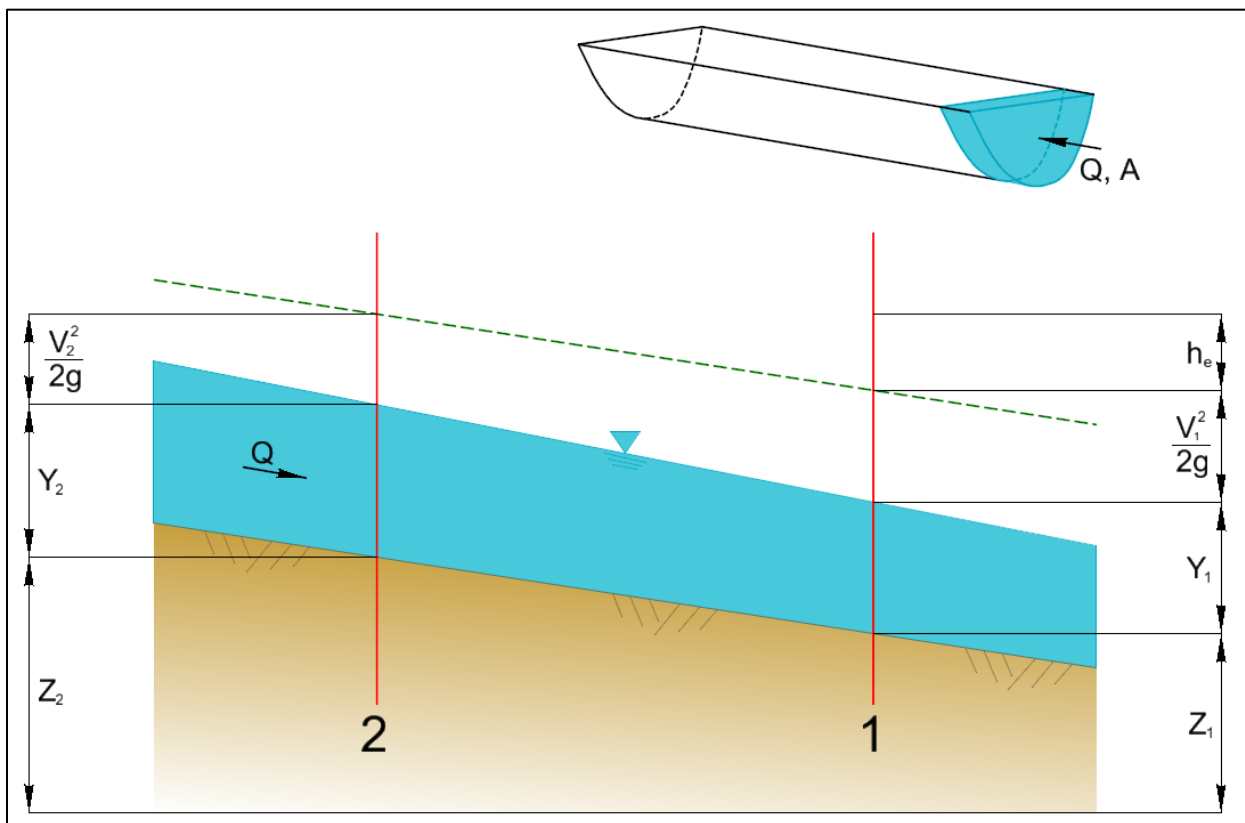
Figure 2.1. Control volume and profile for standard step equation.

2.2.2. Advantages

One-dimensional models have, for decades, been a well-accepted tool for the analysis of non-uniform open channel hydraulics. They have advantages over simple analytical methods (such as direct application of the Manning equation) in that:

- They are not limited to steady uniform flow
- They can handle road crossing hydraulics better than analytical methods
- They can use more extensive geometric data.

Historically, 1D models have been consistently easier and faster to develop and run than 2D models. Although advances in the development of 2D models and associated software have changed this comparison, a 1D model still has certain advantages over 2D models for simple hydraulic situations. These advantages are mostly due to the pervasive use of 1D models and the hydraulic modeling community's procedures for working with them. The advantages include:

- Computational time. Even large 1D steady-flow models can be executed in a short time frame.

- Regulatory convenience. Most regulations and policies relating to waterways (e.g., floodplain regulations) are currently written and administered in a 1D context.

- Design expediency. The formulation of most scour and countermeasure design equations has historically been in a 1D context. While 2D and 3D models produce more accurate (i.e., closer to reality) hydraulic values, 1D models can simplify the process of translating model results into 1D based design equations.

- Portability and familiarity. The HEC-RAS 1D modeling software has been an industry-standard in the US for over 20 years. Computer files and results from HEC-RAS can easily be transmitted, understood, and edited by most current hydraulic engineering professionals.

2.2.3. Assumptions and Limitations

A 1D model is an adequate tool to analyze hydraulic problems that are generally 1D in nature (i.e., flow is in one direction (downstream), constant water surface elevation (WSE) across the waterway, minimal variations in terrain and vegetation), such as an irrigation canal. A 1D model becomes a less reliable tool the more the real-world hydraulics deviate from the 1D assumptions. For a project with a high degree of hydraulic complexity, 1D models can provide extremely erroneous results. The following assumptions are inherent to 1D models:

- Flow Direction. The modeler assumes the flow direction is perpendicular to the user-defined cross sections.

- Flow Path. The modeler assumes the flow path alignment and length along the channel and in the floodplain between cross sections.

- Cross Section Spacing. One-dimensional models assume transverse flow is negligible and essentially "redistribute" flow at each cross section based on the available conveyance area. "Cross section spacing is assumed to be close enough so that the numerical solution to the standard step equation is reasonable. They also assume when flow distribution changes between cross sections, that the flow can physically accomplish the redistribution within the allotted distance (FHWA 2012b)." Physically impossible model results can easily be created by placing cross sections either too close together or too far apart, or by attempting to represent complex, inherently 2D/3D hydraulic situations with a 1D model.

- Flow Splits. Multiple assumptions are required to model flow splits, multiple opening bridges, and flow conditions at confluences. User assumptions include how the model handles the energy loss and WSE at the split as well as the number, direction, location, and discharge amount of each flow split. Lateral momentum is also not accounted for at flow splits.

- Ineffective Flow Areas. The flow is assumed to be perpendicular to a cross section along its entire length and all regions inundated by the computed WSE are assumed to convey flow unless specified otherwise by the modeler with an ineffective flow area.

- Boundary Conditions. One-dimensional models ordinarily require the modeler to place boundary conditions for at least one upstream (inflow) and one downstream (WSE) location. Both boundaries, though primarily the WSE value and location, are potential sources of error due to assumptions made. Additional flow can be introduced at any cross section and is distributed throughout the cross section.

- Flow Contraction and Expansion at Bridges. The modeler assumes how the flow contracts and expands through a bridge, which is controlled by model settings and ineffective flow area placement specified by the user.

- Roughness Parameters. Roughness parameters are averaged at each cross section and assumed to be constant between that cross section and the next.

- Channel Slope Limitation. Channel slopes being modeled need to be relatively "small," generally less than 10 percent.

- Gradually Varied Flow. The flow is assumed to be gradually varied unless specific parameters have been set to calculate rapidly varied flow at specific locations, such as a weir or orifice. HEC-RAS 1D uses the energy equation (conservation of mechanical energy) to calculate the change in WSE from one cross section to the next. This assumption works well for gradually varying flow. However, 1D models do not use the energy equation in rapidly varied flow situations, but instead, use empirical equations and coefficients to make approximations.

- Limited Hydraulic Results. Hydraulic variable results such as WSE, energy grade, velocity, and conveyance are calculated at cross sections and averaged to one value. Modeling software permits several different data types and allows varying levels of detail (e.g., cross section geometry and roughness values). However, input data are eventually averaged into a single value at each cross section. Likewise, multiple values of results can be generated along a cross section by an assumed lateral distribution. However, in both cases, the computations are carried out and produce results that only vary in one dimension. Interpolation of hydraulic variables occurs between cross sections.

2.3. ADVANTAGES, ASSUMPTIONS, AND LIMITATIONS OF 2D MODELING

Two-dimensional modeling represents a significant advancement over 1D modeling by allowing flow conditions to vary in two dimensions instead of one. They have fewer internal assumptions and require significantly fewer assumptions by the modeler. Two-dimensional models calculate hydraulic results at locations within a mesh that covers the entire geographic extents of a river and floodplain. For non-hydraulic audiences, 2D models provide a more intuitive graphical representation of results as opposed to cross sections and tables from a 1D model. However, 2D models are bound and limited by the assumptions made, internal to the model and by the modeler, to produce a manageable computational problem.

Many terms introduced in this section are expanded on in later chapters. For an explanation of computational terms, see Chapter 3. For an explanation of common model terms, see Chapter 5.

2.3.1. Depth-Averaged Models

Two-dimensional models assume an average velocity distribution in either the lateral or vertical (depth) dimension. These internal simplifications significantly reduce the numerical complexity of the resulting problem, producing more manageable numerical problems compared to 3D models

or reality. Most 2D models used in water resources and hydraulic engineering are depth-averaged velocity models.

Depth-averaged velocity models assume that velocity in a given control volume is averaged in the vertical (z) dimension and varies in the longitudinal and lateral (x,y) directions. The focus of this reference document is on the application of 2D depth-averaged velocity models for transportation design and analysis problems. Section 2.4 presents FHWA's recommended minimum and extended 2D model capabilities for transportation design problems. Section 3.4 presents one representation of the shallow water equations (SWE) solved by depth-averaged 2D models, the simplifying assumptions made by the SWE, and the limitations associated with them.

For this section, it is helpful to understand the additional processes that 2D models account for compared to 1D models. Most 2D models use a variation of the depth-averaged 2D SWE to solve for the flow variables at each location. The SWE accounts for the many nuances of flow such as flow direction, velocity distribution, momentum, turbulence, diffusion, and their gradients in the x and y directions. The methods used to compute each of these components of flow determine many of the limitations of the specific model used.

2.3.2. Advantages

Many of the advantages of 2D models pertain to flow-related characteristics solved for by 2D models but are specified or entirely neglected by traditional 1D models. A hydraulic analysis using a 2D model requires fewer physical simplifications and user-specified parameters. Consequently, a 2D model can provide additional insight with fewer user assumptions relative to even a well-posed 1D model (Table 2.1). These computational advantages include:

- Computed Flow Direction. A 2D model directly calculates flow path and direction. A physics-based calculated flow path and direction provides an enormous advantage, especially for applications such as the angle of attack for bridge scour analyses. Two-dimensional models can resolve lateral, multi-directional, and even recirculating flow patterns.

- Varying Water Surface Elevation. Two-dimensional models can solve for longitudinally varying water surfaces, which allows the modeler to determine the depth of flow, both in the channel and within the floodplain. This level of spatial resolution and detail can be significant when analyzing bridge hydraulics, extents of flooding, designing channel training structures and countermeasures, or when evaluating potential flood mitigation solutions.

- Flow Distribution. The application of the full SWE allows a 2D model to solve for complex flow distributions. A momentum-based 2D model can track the distribution of flow at flow splits, multiple openings, across wide floodplains, through multiple channels, around bends, and across lateral weirs.

- Turbulence, Flow Recirculation, and Ineffective Flow Areas. Two-dimensional models can provide realistic hydraulic representations of the flow conditions at various resolutions throughout the model domain, allowing the determination of flow separation, recirculating locations, and backwater areas. For 1D analysis, the modeler makes an assumption and informs the model where such locations exist. This capability can be extremely useful when evaluating countermeasure placement, understanding areas of ineffective flow, or analyzing navigability issues.

Table 2.1. 1D versus 2D modeling

Hydraulic Variables	1D Modeling	2D Modeling
Flow direction	**Assumed** by user	**Computed**
Flow paths	**Assumed** by user	**Computed**
Channel roughness	**Assumed** constant between cross sections	Roughness values at individual elements used in computations.
Ineffective flow areas	**Assumed** by user	**Computed**
Flow contraction and expansion through bridges	**Assumed** by user	**Computed**
Flow velocity	**Averaged** at each cross section	**Computed** at each element
Flow distribution	**Approximated** based on conveyance	**Computed** based on continuity and momentum
Water Surface Elevation	**Assumed** constant across entire cross section	**Computed** at each element

2D models also excel when it comes to the ability to represent and analyze waterways spatially. Software applications used to develop most 2D models have similar capabilities to geographic information system (GIS) applications. The integration of spatial features creates many advantages, including:

- Realistic Spatial Attributes. A modeler can include geospatial data such as aerial images, topography, infrastructure data, land use data, and design information directly into the 2D modeling environment. These data types can be translated directly into model components such as mesh structure, terrain elevations, surface roughness, and upstream and downstream locations. While the development of many 1D models now occurs within a geospatial context, there is a difference between interpolating average values as required in a 1D context and the ability to more completely represent spatial variables within 2D model computations.

- Data Visualization. Profiles, cross sections, and data tables are familiar to engineers who have used 1D models, but these data can be challenging to interpret and understand for some less experienced stakeholders involved in a project. Two-dimensional models produce results that can be projected onto maps and displayed in a wide range of colors and geometric schemes (e.g., plan or oblique views). Intuitive display of hydraulic results can be a transformative advantage that can assist engineers in communicating hydraulic nuances to a variety of audiences.

- Highly Refined Numerical Resolution of Flow Fields. Numerical models provide hydraulic results at computational points. The computational point is either the center of an element or each mesh node (the points combining the mesh element edges). A 2D model with variable mesh resolution allows the modeler to increase the resolution in highly variable regions of flow or areas of design interest by reducing the element size. By comparison, a 1D model only computes hydraulic results at cross-sections, and any lateral information or conditions away from the cross-section are post-processed or interpolated.

2.3.3. Assumptions and Limitations

Two-dimensional models have a very different set of physical assumptions and limitations compared to 1D models. One-dimensional models neglect many aspects important in hydraulic analyses of river systems. The following assumptions are typically involved in 2D modeling:

- Dimensional Limitations. Depth-averaged 2D models assume that variables change spatially in only the horizontal plane, which means that 2D models are not able to resolve vertical variations in hydraulic flow properties. This results in the following assumptions:

  - There is no vertical component to velocity vectors.
  - There is no vertical component of flow diffusion.
  - There is no vertical component of flow turbulence.
  - Vertical velocity distribution is assumed to conform to a standard distribution (see Section 3.2)

- Boundary Conditions. Similar to all models, 2D models require the modeler to place boundary conditions at all open boundaries, ordinarily for at least one upstream (inflow) and one downstream (outflow or WSE). Establishing these boundaries can create potential sources of error due to assumptions made at each boundary. Section 5.4 provides more detail related to boundary conditions for 2D models. Section 6.4 includes information related to sensitivity analysis related to boundary conditions.

- Wetting and Drying Limitations. For a 2D model to perform flow computations at an element, it first determines if the element is inundated (wet) or dry. Different models use different wetting and drying schemes. To produce results, some models require that all nodes of an element be wet. Other models calculate a representative average ground elevation for an element and then evaluate inundation based on that elevation. Others incorporate various approaches to allow elements to be partially wet. Depending on the terrain and mesh development, a model may produce unstable or errant results along the model boundary due to wetting and drying of elements. Newer generation models account for wetting and drying with a significantly more stable approach than older generation models. Also, newer generation models may incorporate post-processing features to help map a representative wet/dry boundary.

- Accounting for Turbulence. Many 2D models include the depth-averaged turbulent stresses. To appropriately model turbulence, the model considers the transfer of momentum between elements. The most common method used to approximate these internal momentum transfers is through the use of eddy viscosity. In highly turbulent regions, treating the water as a more viscous fluid may be necessary to maintain numerical stability.

  - How a 2D model calculates and applies eddy viscosity is an assumption and limitation of the specific model used. Models usually provide the modeler with options to adjust this calculation to suit the needs of a particular problem. The modeler should be aware that over- or under-estimating eddy viscosity may result in unrealistic flow patterns or model instability. These effects may be most evident in highly turbulent regions.

- Wall Effects Assumption. 2D models are capable of modeling the edge of inundation as a "no-slip" or "slip" boundary. A no-slip boundary assumes flow velocity is zero at the boundary, and a slip boundary allows the calculation of a velocity value at the boundary. A no-slip boundary has a lateral zone of influence with a variable velocity distribution approaching the boundary.

2.8

The methods used to approximate the velocity distribution associated with a no-slip boundary differ from one model to another and can be affected by model configuration properties such as mesh density, terrain, and roughness.

- Neglected Terms. Some 2D models neglect certain terms in the SWE to simplify the numerical complexity of the model simulation. The most commonly deleted terms include inertial (acceleration) and pressure terms. Such models produce less useful results when resolving turbulent flow fields in complex hydraulic situations. Modelers should verify what terms are being used in a 2D model and determine if that model is sufficient for what they are analyzing. FHWA recommends the use of 2D models that incorporate the full dynamic wave equations for transportation-related hydraulic analyses. Section 3.6 provides a discussion of model simplifications in more detail.

- Model Stability and Convergence. All numerical models, especially 2D models, require a stability and convergence check. If these issues arise, the model may fail to run, may diverge, or may produce errant results that may appear realistic. In other cases, the model could produce unstable results, such as oscillating water surface elevations. Section 6.1 provides stability and convergence definitions and recommendations to detect and correct potential issues.

2.4. 2D MODEL SELECTION FOR BRIDGE AND CULVERT HYDRAULICS

The number of depth-averaged 2D models available to the modeler has increased as computational capacity and numerical efficiency improves with successive generations of software and hardware. Every model is designed and optimized for a subset of potential hydraulic modeling problems and design applications. In addition to selecting a 2D model based on its capabilities, users should also evaluate the model's ease of use along with its continued development and level of technical support. This section presents recommended model capabilities for typical bridge and culvert hydraulic design problems. Chapter 3 presents the underlying computational assumptions and numerical characteristics underpinning these recommendations.

At a minimum, FHWA recommends that any 2D model used for bridge hydraulic design problems have the following capabilities:

- Able to perform depth-averaged computations of velocity components in the horizontal (x-y) plane and WSE.

- Able to ensure conservation of mass. Different model schemes conserve mass either exactly or approximately at different scales. See Chapter 3 for additional discussion on mass conservation. Flow continuity, and sediment continuity if applicable, should be assessed and verified as part of model review.

- Able to conserve momentum in the horizontal plane.

- Able to accommodate unsteady flow conditions (time-varying boundary conditions and solutions).

- Capable of directly modeling hydraulic structures in the model domain (direct feature representation), either as part of the mesh or with a linked model. FHWA recommends direct feature representation in the mesh if the model can resolve the hydraulics given the model's underlying assumptions. Hydraulic structures may include:

2.9

- Bridge elements (piers, abutments, bridge decks, countermeasures, if required)
- Culverts
- Weirs
- Gates
- Storm drains

Depending on the specific design application, FHWA recommends the following optional model capabilities for transportation-related hydraulic design:

- If hydraulic conditions are influenced significantly by wind effects (such as for crossings of large exposed bodies of water, coastal estuarine models, or coastal regions directly exposed to landfalling storms), the model should account for wind shear on the free WSE boundary of the model. See "A Primer on Modeling in the Coastal Environment" (FHWA 2017) and HEC-25 "Highways in the Coastal Environment" (FHWA 2008) for additional information.

- If hydraulic conditions are impacted substantially by interaction with rainfall or groundwater (infiltration or exfiltration), the model should account for these additional mass inputs.

- If sediment transport analysis is necessary, the model should perform mobile bed simulations. In this case, it is also desirable that the mobile-bed model account for geotechnical stability of underlying soils while conserving sediment continuity. See Section 8.4 for additional discussion of mobile-bed hydraulic analysis.

FHWA also recommends the use of unstructured meshes (i.e., non-uniform element sizes) for bridge hydraulic analysis and design purposes, as described in Section 5.3. Unstructured meshes allow the computational mesh configuration, orientation, and scale to vary within the model domain. By using this type of mesh, a modeler can develop a mesh that represents all hydraulically significant topographic features (see Section 4.2) and adequately resolves the flow field in regions of design interest while maintaining reasonable computational efficiency. See Chapter 5 for an extended discussion of unstructured mesh development.

Table 2.2 presents a partial list of 2D depth-averaged hydraulic models showing model capabilities. For the hydraulic structure capabilities, the table represents the model's capabilities as they relate to modeling transportation infrastructure. For example, there are several methods for modeling piers; some methods are adequate for modeling applications relative to bridge design, and some are not. This list is not all-inclusive and does not imply FHWA endorsement of the listed models for any particular design application. All modeling capabilities are as reported in the model's corresponding support documentation at the publication date of this document.

Table 2.2. Various 2D model capabilities related to transportation hydraulics.

Model	Basic Capabilities					Hydraulic Structures[1]					Other Capabilities			
	Unstructured Mesh	Conservation of Mass	Conserves Momentum	Supercritical Flow	Unsteady Flow	Bridge Piers[2]	Bridge Pressure Flow	Weirs[3]	Culverts[4]	Gates	Wind Shear	Rainfall	Infiltration & Exfiltration	Mobile Bed
ADCIRC	X	X	X		X	X		X	X	X	X			X[5]
ADH	X	X	X	X	X	X	X	X	X	X	X	X	X	X
FLO-2D		X	X[6]		X			X	X			X	X	X
Hydro-AS	X	X	X	X	X	X	X	X	X	X	X	X	X	X
Mike21C	X	X	X	X	X									X
HEC-RAS-2D	X	X	X[7]	X	X			X	X	X		X		
River FLOW2D	X	X	X	X	X				X		X	X	X	X
SRH-2D	X	X	X	X	X	X	X	X	X	X		X	X	X
TELEMAC	X	X	X	X	X	X					X			X
TUFLOW- (FV & Classic)	X	X	X	X	X	X	X	X	X	X	X	X	X	X

X = has capability

[1] Some models can represent hydraulic structures as 1D/2D features; refer to manuals for further information.

[2] Piers that are explicitly represented as holes in the mesh or using other direct methods that have been verified to provide detailed hydraulic results through the bridge opening.

[3] Models that can resolve sub/super/trans-critical flow and use unstructured meshes that adequately resolve broad-crested and Ogee spillways as 2D features, some models can also represent weirs with 1D features.

[4] Box culverts can be represented as a combination of mesh voids and bridge deck features; some models can represent culverts with 1D features.

[5] Not in public version

[6] 8-direction mass and momentum conservation only

[7] Only when "Full momentum" method is selected; default is diffusive wave (see Section 3.6.1)

2.5. FUTURE OF 2D MODELING AND OTHER ADVANCED HYDRAULIC MODELING

User-friendly and numerically stable models have become increasingly widespread. FHWA has committed to working with applied scientists, the engineering consulting community, regulatory stakeholders, and infrastructure owners to develop and implement more efficient and effective procedures and design tools. These 2D modeling procedures and tools are being optimized to provide additional insights and improved representations of river systems that 2D modeling brings to hydraulic design and analysis.

Engineers and designers who are accustomed to the use of 3D spatial data sets in the GIS/CAD environment are able to take advantage of these improved tools to speed production, increase

design consistency, and simplify review. Basic engineering curricula now commonly includes training in spatial processing, and engineering application of surface processing tools is common practice in many disciplines.

2.5.1. 2D Modeling Improvements

In the short term, FHWA has identified 2D modeling development needs to address common design problems, including:

- Floodplain Analysis. Additional resources are needed for working with federal, and regional, floodplain regulations including delineating floodways and no-rise evaluation. FHWA has engaged the floodplain administration regulatory community in advancing the use of these tools to support the interactive development of regulatory procedures and standards that reflect 2D model insights and capabilities.

- Scour analysis. Historically, many of the design procedures used to evaluate scour have been simplified to use the results obtained from rigid bed 1D modeling. Two-dimensional models provide better representations of sediment-transport conditions at a given location and can provide more insight into scour conditions at a bridge. Ongoing scour research performed by FHWA seeks to take advantage of enhancements to existing scour equations and allow 2D model results to simplify analysis, better quantify and reduce uncertainty and unnecessary conservatism, and reduce design uncertainty.

Long-term developments in 2D modeling include using mobile-bed hydraulic principles to evaluate contraction and local scour directly. This approach requires advances in relating 2D model results to observed scour data, including:

- Reviewing existing prediction methods and developing training for design practitioners in the application of 2D mobile-bed hydraulic models to engineering design problems.

- Developing prediction methods to evaluate the difference between 2D mobile-bed results and available research data sets.

2.5.2. Advanced Modeling Tools

Some complex hydraulic problems are beyond the ability of 2D models. Advanced modeling options include different types of 3D modeling and physical modeling in a laboratory. It is beyond the scope of this document to describe advanced modeling in detail or to describe their application to advanced engineering practice. HDS-7 presents additional details in determining when advanced modeling tools may be necessary for a given modeling task.

Three-Dimensional Modeling. Three-dimensional modeling commonly referred to as "computational fluid dynamics" or CFD, does not assume depth-averaged flow. Three-dimensional models can resolve flow in the z-dimension and calculate vertical velocity gradients within the vertical resolution of the model (FHWA 2012b). Three-dimensional models are currently a "state-of-the-art" engineering tool, but they could be of significant value in regions with highly stratified flows or where a detailed understanding of flow patterns would be valuable. CFD modeling may be warranted if 2D modeling assumptions cannot adequately resolve hydraulic conditions necessary for design. If hydraulic conditions include significant vertical accelerations or vertical velocity components that are significant to the analysis, then a 3D model should be considered. Some examples include, but are not limited to, analysis of detailed circulation fields in reservoir diversion structures and forebays, spillway hydraulics, flow patterns near flow

obstructions (e.g., barbs, piers, etc.), non-standard complex pier configurations, or scour patterns at engineered wood structures.

Historically, the computational intensity of 3D modeling and the lack of modeling tools tailored to hydraulic analyses and civil engineering have limited applications to research and very detailed and small-scale engineering problems. As of 2019, bridge-design scale 3D modeling still requires specialized computational tools or investment in cloud computing resources. As computing power advances 3D models will transition from specialized applications, primarily used in a research context to extend the results of 2D and physical model studies, to become one of the hydraulic modeling approaches commonly available to the practitioner.

Lagrangian 3D Models. Most hydraulic models use equations of motion and continuity that resolve forces acting on a control volume (see Section 3.2), known as Eulerian models. Another form of 3D modeling tracks the interactions and positions of individual fluid and other particles rather than tracking the mass entering and exiting a control volume. These are known as Lagrangian models. A Lagrangian approach can model representative particles of different sizes and properties, such as air, water, and sediment. Application of Lagrangian models is already the state-of-practice in many engineering fields, including aerospace, industrial process engineering, and computer-generated imagery for movie special effects and video games. Currently, the application of Lagrangian models to hydraulic and sediment transport problems only apply to small-scale research analysis. As data collection and computational capabilities improve, the application of Lagrangian models may extend to additional engineering fields.

Physical modeling. Physical models represent a river reach of interest with a simplified and (usually) smaller-scale physical representation (Julien, 2008). Like 3D numerical models, physical models can provide insight into vertical velocity profiles and, at larger-scaling factors, quantitative insight into complex flow patterns necessary for design or forensic analysis scenarios. However, complex scaling interactions of geometry, flow dynamics, sediment fall-velocity (settling velocity for sediment particles), and incipient motion can increase the complexity and limit the validity of physical modeling results (Julien, 2008). Physical modeling can be labor-intensive and requires specialized facilities and equipment, including a flume, access to water and sediment, labor, and machine tools to construct a physical representation of the river reach of interest. Physical modeling, similar to numerical modeling, requires specialized training to develop an appropriate experimental design that captures the data necessary to quantify experimental results. A common analytical approach is to extend physical model study results with calibrated 2D and 3D numerical model analyses.

CHAPTER 3

2D MODELING FUNDAMENTALS

This chapter builds on model assumptions discussed in Chapter 2 and describes the governing equations, standard numeric approaches, and simplifying assumptions associated with those approaches. This chapter does not attempt to provide a detailed description of numerical methods or theory.

3.1. PHYSICAL PRINCIPLES

It is important to understand the physical principles represented in a typical model. Most models include aspects of both fluids at rest and in motion. Forces associated with liquids at rest include buoyancy and density that create pressure on dams, embankments, culverts, and other structures. In contrast, forces associated with liquids in motion include flow resistance due to friction and turbulence generated by fluid flow.

Key physical properties and parameters used to describe and quantify fluid motion include acceleration, velocity, mass, momentum, and energy:

- Acceleration is the rate of change of velocity with respect to time.

- Velocity is the speed of an object in a given direction. Velocity is a vector quantity; both direction and magnitude are important.

- According to Newton's 2nd law of motion, a net force applied to a mass will cause the mass to accelerate at a constant rate in the direction of the applied force.

- Mass is the measure of an objects (body) resistance to acceleration.

- Weight is the product of the mass of an object and the local value of gravitational acceleration.

- Momentum is the product of the mass of an object and its velocity.

- Mechanical energy is the sum of potential and kinetic energy.

These physical properties are related to each other in the following ways:

$$F = ma \tag{3.1}$$

$$M = mv \tag{3.2}$$

$$E = \frac{m*v^2}{2} + m*g*d \tag{3.3}$$

where:

$$
\begin{aligned}
F &= \text{force} \\
m &= \text{mass} \\
a &= \text{acceleration} \\
M &= \text{momentum}
\end{aligned}
$$

v   =   velocity magnitude
g   =   acceleration due to gravity (constant)
d   =   distance/depth above datum
E   =   mechanical energy

Using these relationships and the fundamental principles of the conservation of mass and momentum, the Reynold's averaged Navier Stokes (RANS) equations (described in the following section) have been derived from Newton's equations of motion. The RANS equations provide the basis to perform numerical hydrodynamic analysis. These equations include unknowns of water level and velocity. Velocity is a vector quantity, so in two dimensions, this represents two unknowns. A 2D numerical representation of a flow domain, therefore, has three unknown values for each computational location. The computational points can be mesh nodes, element centroids, or special locations inside each element, depending on the formulation. The numeric implementation compiles these three unknowns for each location into a system of equations that are solved simultaneously.

When applying these numerical methods, additional constants (or conditions) representing the flow should be provided by the user to define the flow field. This information is often referred to as the constants of integration in theory or boundary conditions in numeric theory. In hydraulics, the modeler specifies flow values (water level and velocity) at all edges of the domain. This boundary definition allows the numeric solver to compute the values at all computational locations.

To simplify what should be specified by the modeler, most models assume that the boundary flow is zero unless otherwise specified. Therefore, the user specifies boundary conditions only for locations where flow enters or leaves the model. In a riverine system, this is typically the upstream and downstream ends of the simulation. These boundary conditions include water levels and flow rates. Depending on the specific numeric implementation, the specified flow rates are converted to velocities at the boundary elements using either input from the user or assumptions about flow direction and distribution.

How the numerical system works can be visualized by thinking of a global mass balance calculation. If the flowrates of water entering and leaving a system are defined, it is possible to compute the total volume of water in the system at any point in time by adding the cumulative inflow and subtracting the cumulative outflow. The model computes this mass balance for each element in the mesh. Depending on the numerical method being employed (see Section 3.2), this conservation can be enforced locally or approximated locally and enforced globally. Some models also allow mass to enter or leave through the bed, water surface, or internal boundaries. Mass entering or leaving through the bed represents infiltration or exfiltration to and from groundwater storage. Mass entering or leaving through the water surface represents rainfall and evaporation.

The modeler also specifies parameters to represent forces that act on the water mass in each element. The primary forces include friction at the bed, friction at the water surface, and pressure. Friction at the bed is quantified through specified roughness values (usually Manning's N values). Friction at the surface is quantified using specified wind speeds. Pressure is derived iteratively from the computed water levels. These forces are represented as partial differential terms in the equations for the conservation of momentum

3.2. GOVERNING EQUATIONS FOR DEPTH-AVERAGED 2D MODELS

The Navier-Stokes equations, named for Claude-Louis Navier and George Gabriel Stokes, apply simplifying assumptions to represent motion of fluids. Most 2D models use a variation of the depth-averaged SWE derived from the Navier-Stokes equation. Different numerical models apply a specific derivation and representation of these equations that form the basis of the formulations necessary to perform the numerical analysis. The depth-averaged velocity principle discussed previously is shown in Figure 3.1. A depth-averaged model most commonly assumes a logarithmic velocity distribution within the water column.

$$U = \frac{1}{H} \int_{Z}^{Z+H} u \, dz$$

(3.4)

where:

U, V	=	depth-averaged velocity (U corresponds to the x-direction, V to y)
H	=	depth of water
Z	=	elevation of channel bottom
u	=	velocity of water at corresponding depth of water z
z	=	depth of water at corresponding velocity u
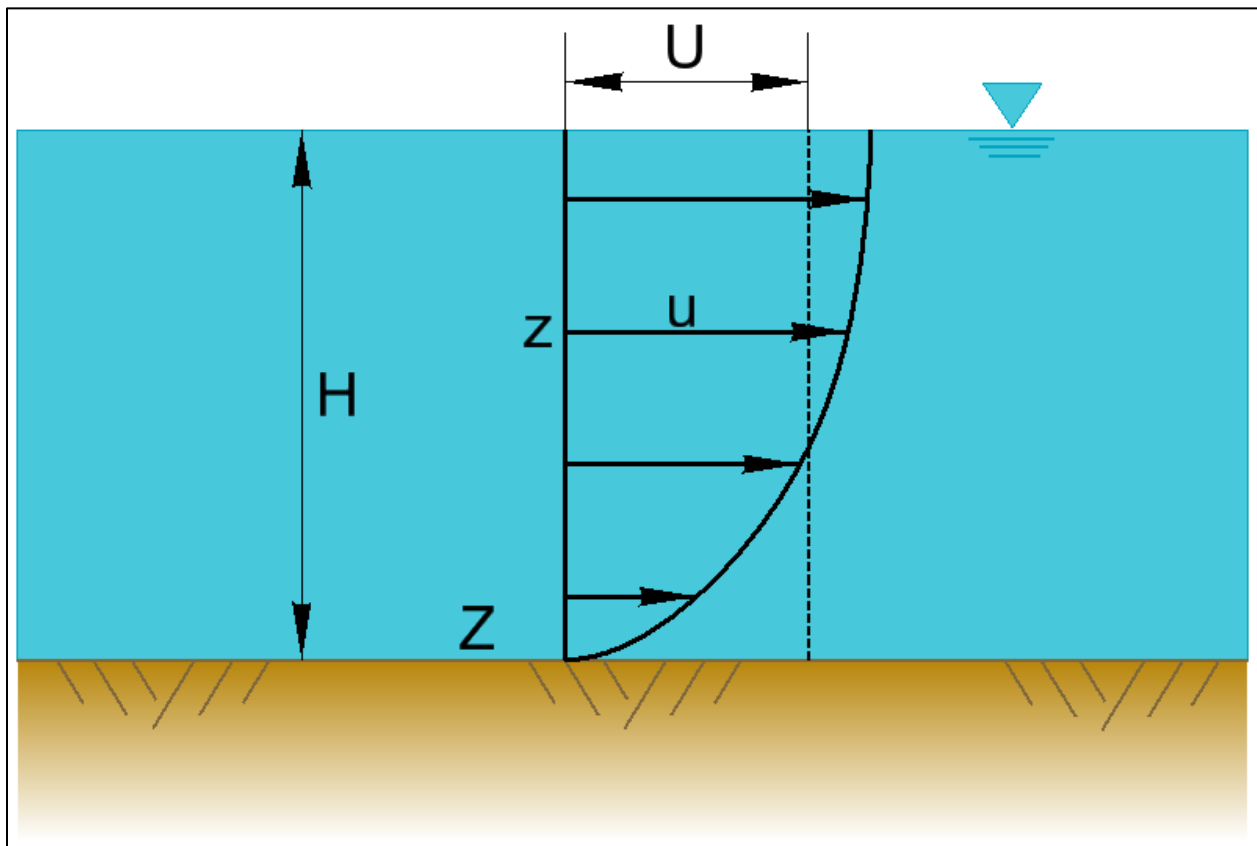
Figure 3.1. Depth-averaged velocity principle.

The 2D approach can be communicated visually by representing the attributes of a finite column of water, as displayed in Figure 3.2.
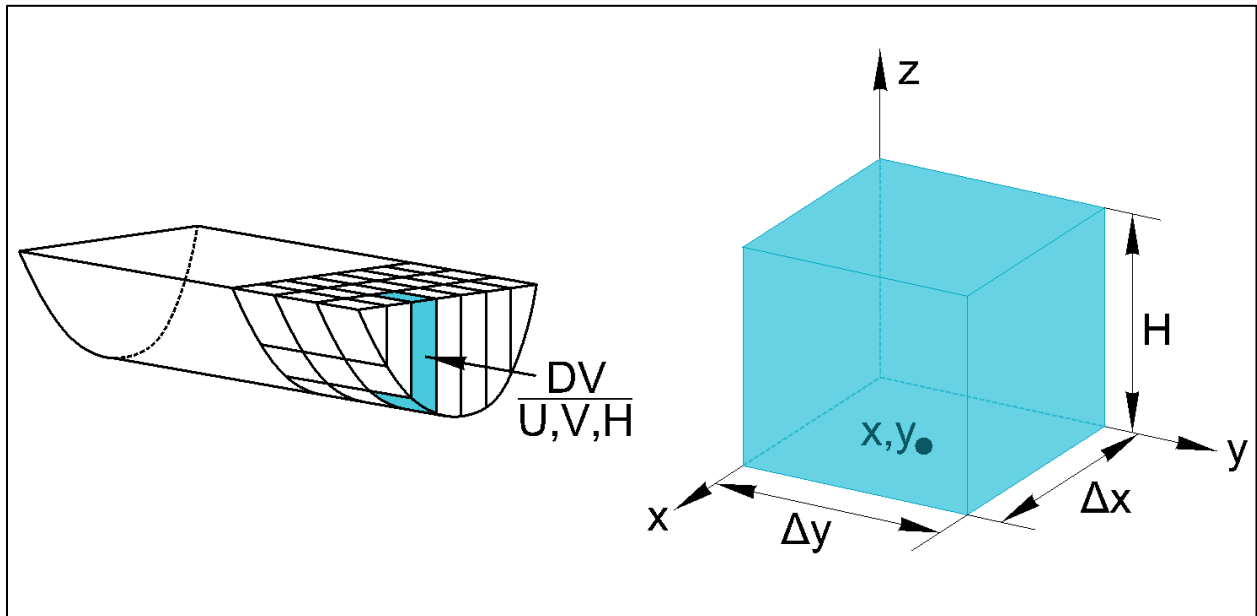
Figure 3.2. Two-dimensional approach, example water column.

The SWE consider the continuity and momentum principles (Figure 3.3). The continuity principle (conservation of mass), accounts for flow into and out of all six sides of the water column and requires that the rate of change of mass within the water column equals the net flux of mass entering the column. The total flow into the water column equals the total flow out, plus any change in storage.
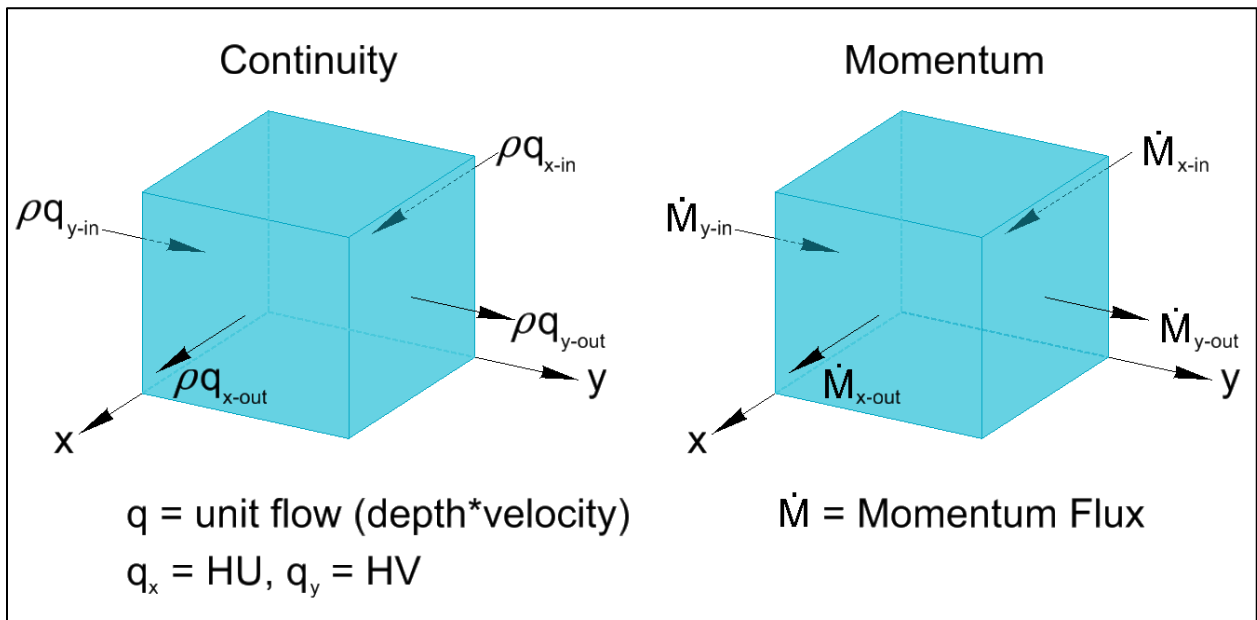
Figure 3.3. Continuity and momentum principles.

The momentum principle requires that the rate of change of momentum within the column minus the rate at which momentum flows into the column plus the rate at which momentum flows out of the column should be equal to the sum of forces acting on the column. Inertial terms related to the momentum of the fluid can be included. The net change in momentum in the control region is usually expressed in the momentum equation as the term $\rho(du/dt)$ or $(\rho v)v$.

The various forces acting on the column include the body forces (gravity and forces related to the rotation of the earth know as Coriolis forces) and the surface forces, including hydrostatic, bed shear, wind shear, and turbulence as shown in Figure 3.4. Representation of turbulence can include dispersion terms that account for mixing of water. Different parts of the fluid flow at different rates, bringing into proximity fluid mass from different areas and with different momentum that represents non-uniform motion of particles in the water column. The conservation of mass and momentum through the domain ultimately produce a set of PDEs of the SWE that need to be solved within the domain to compute the 2D hydraulics. The SWE account for the many nuances of flow shown in Figure 3.5.
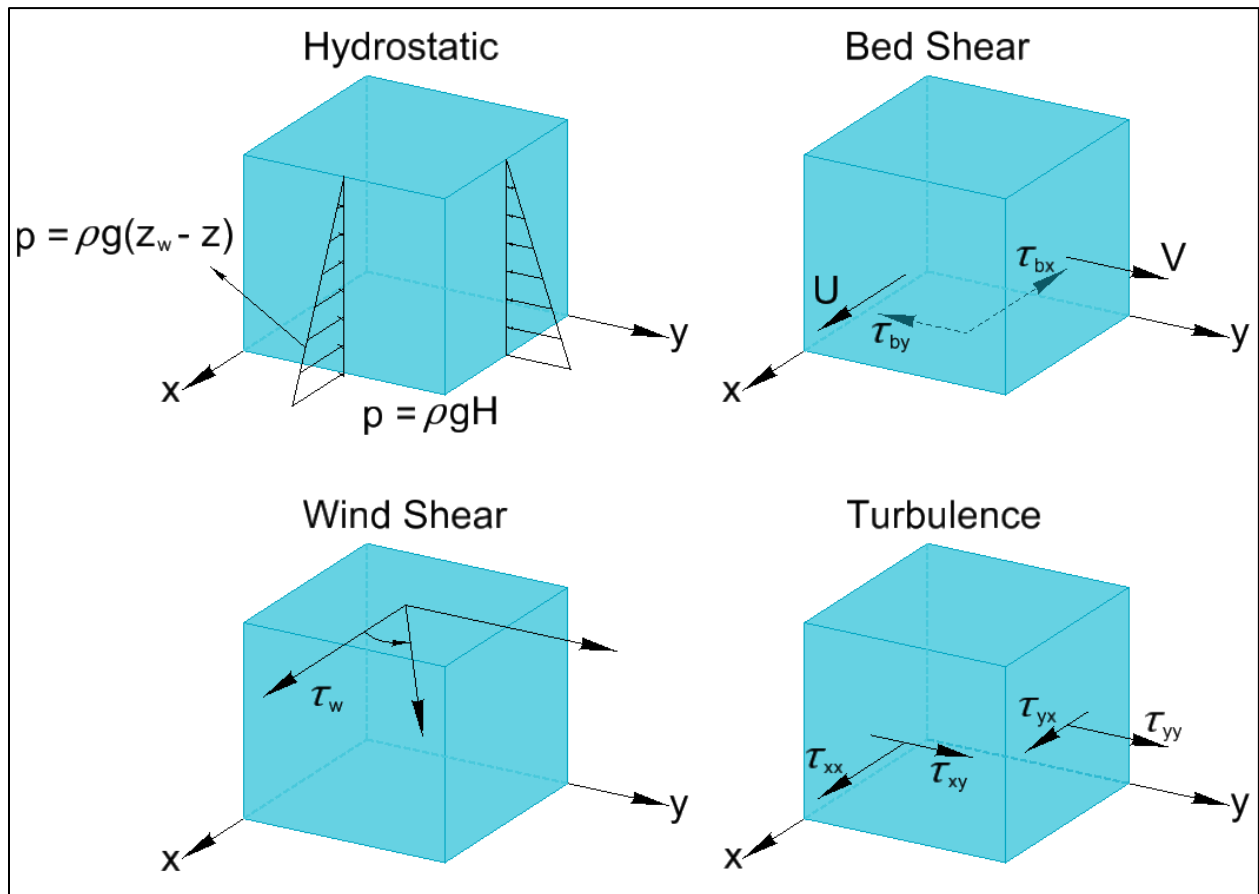
Figure 3.4. Surface forces acting on a water column.
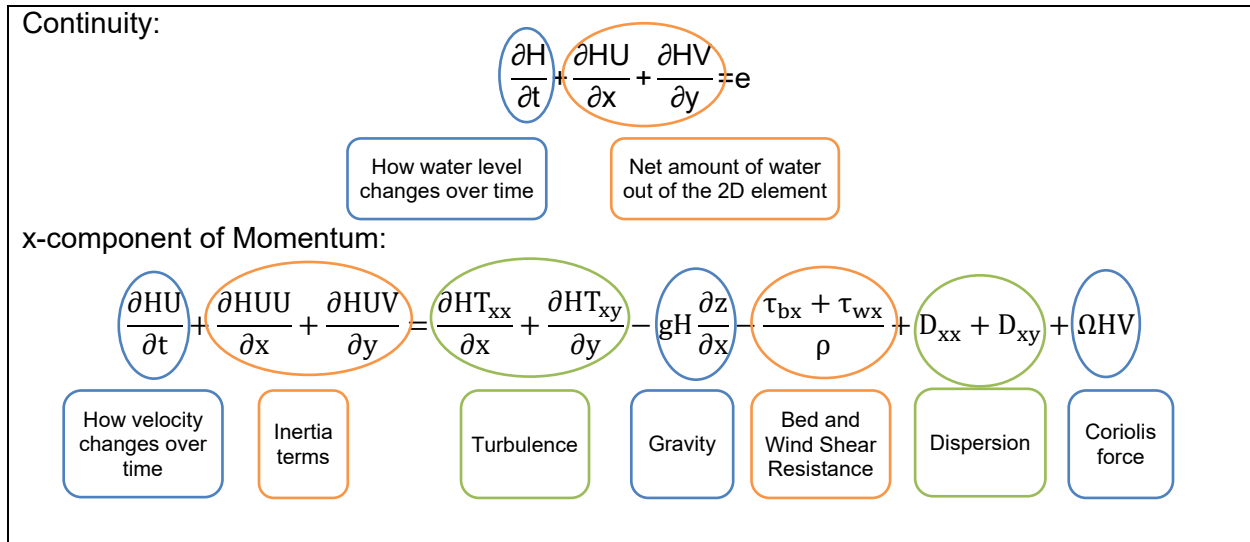
Figure 3.5. Example 2D continuity and momentum equations, and their contributing terms.

As noted previously, different models use different variations of the SWE. The momentum equations presented below are one typical example:

The continuity equation is:

$$\frac{\partial h}{\partial t} + \frac{\partial HU}{\partial x} + \frac{\partial HV}{\partial y} = e$$

(3.5)

The momentum equation in the x and y-direction is:

$$\frac{\partial HU}{\partial t} + \frac{\partial HU^2}{\partial x} + \frac{\partial HUV}{\partial y} = \frac{\partial HT_{xx}}{\partial x} + \frac{\partial HT_{xy}}{\partial y} - gH\frac{\partial z}{\partial x} - \frac{\tau_{bx} + \tau_{wx}}{\rho} + D_{xx} + D_{xy} - \Omega HV$$

(3.6)

$$\frac{\partial HV}{\partial t} + \frac{\partial HV^2}{\partial y} + \frac{\partial HVU}{\partial x} = \frac{\partial HT_{yy}}{\partial y} + \frac{\partial HT_{xy}}{\partial x} - gH\frac{\partial z}{\partial y} - \frac{\tau_{by} + \tau_{wy}}{\rho} + D_{yy} + D_{yx} + \Omega HU$$

(3.7)

where:

x, y	=	Cartesian coordinates
t	=	time
H	=	depth of water
U	=	depth average velocity of water in the x-direction
V	=	depth average velocity of water in the y-direction
e	=	net mass input (rainfall + exfiltration – infiltration)
z	=	WSE (bed elevation + h)
g	=	gravitational acceleration
$T_{xx, xy}$	=	depth average turbulent shear stresses
$D_{xx, xy}$	=	dispersion terms
$\tau_{bx, by}$	=	bed shear stress
$\tau_{wx, wy}$	=	wind shear stress
Ω	=	Coriolis parameter

3.3. MODELING APPROACHES

Many approaches have been developed to represent a systematic numerical method for solving PDEs. These methods generally fall into one of three categories: finite difference, finite element, and finite volume. The approaches subdivide (discretize) the model domain into small geometrically simple shapes. These shapes are referred to as elements (or cells). The collection of elements constitutes a mesh (or grid) that represents the model domain. If the elements are all uniform shapes and sizes (i.e., squares or rectangles) common terminology refers to it as a structured grid. The term unstructured grid applies when the set of elements includes variable sizes and shapes. The term mesh is another name for an unstructured grid.

3.3.1. Finite Difference

The finite-difference method is the most direct approach to mapping differential equations to individual elements in the domain. This method replaces the representation of the equations as a continuum with a set of piecewise equations (finite-difference equations) at a series of points in the domain space. This method typically operates on a grid that capitalizes on the regular spacing of points to improve the efficiency of the solution method. However, this dependence on a grid renders the approach less applicable for irregularly shaped domains (e.g., river channels). Key points relative to the finite-difference method include:

- Computation locations are typically the center of each cell.

- Mass is conserved at each cell; therefore, the method conserves mass globally.

3.3.2. Finite Element

- The finite-element method takes the continuous field equations that describe the physics of hydraulics (as defined above) and formulates these equations for each element. This is handled by approximating the fields within each element as a simple function such as a linear polynomial with a finite number of degrees of freedom. This approach provides an approximate localized description of the physics of flow in the element by a set of linear equations. By assembling the contribution of all the elements, the finite element method results in a large matrix with one row for each unknown in the system (3 unknowns for each computation point). However, since the unknowns for a given element are only influenced by the immediate neighbors of that element, the majority of the terms in each equation is zero. This is referred to as a large sparse matrix equation system, which can be solved using well-known matrix solvers.

The finite-element method is very general. It has been applied successfully to many physical problems and can easily adapt to higher-order elements (quadratic, cubic, etc.) rather than being limited to only linear shapes. These higher-order elements increase the precision of the representation of the unknowns in an element. Key points relative to the finite-element method include:

- Computation locations are special locations inside each element commonly referred to as Gauss points.

- The method iterates locally on all elements to conserve mass globally. It conserves mass approximately at the element scale.

3.3.3. Finite Volume

The finite-volume method is similar to the finite-element method by using a mesh; however, numerically, the basis of the finite-volume method varies significantly from the finite-element method. In contrast to the finite-element method, the finite-volume approach capitalizes on the fact that many physical laws are similar to the conservation of mass and the conservation of momentum principles described above. Based on this concept, the approach builds a formulation that consists of flux conservation equations defined in an averaged sense over the elements.

The finite-volume method's strength is that it only needs to perform flux evaluations for the element boundaries. This strength also holds for nonlinear problems, which makes the approach powerful for nonlinear conservation laws that frequently appear in transport problems (i.e., sediment transport or water quality). Mesh refinement can be used to capture small-scale hydraulic variations similar to the finite-element method. However, because the finite-volume method relies only on conservation across edges, it does not benefit from increasing the order (i.e., quadratic or cubic) of the elements. Key points relative to the finite-volume method include:

- Computation locations are typically the center of each element or the center of each face between elements.

- The method identically forces conservation of mass in all elements; therefore, it conserves mass locally and globally.

3.4. MODEL ASSUMPTIONS

Statistician George E. P. Box is credited with the statement that "All models are wrong: some models are useful." This statement refers to the fact that a model is only an approximate representation of reality and, therefore, does not match reality perfectly.

Scientists and engineers have defined 2D models with simplifying assumptions to allow for mathematical solutions. An understanding of these assumptions allows the modeler to ascertain the applicability of a model to a specific scenario or set of conditions. Section 2.3 and 3.2 discuss common numerical assumptions and present the depth-averaged assumption. When the physical conditions being represented by the model do not conform to the assumptions made in a model's formulation, the model is more "wrong" and less "useful."

3.4.1. Approximated Surface Assumption

All numerical models approximate the underlying topography of the domain of interest into computational nodes (see Section 5.3). For example, 1D models only compute hydraulic conditions at a point represented by a user-defined cross-section, whereas 2D models compute hydraulic conditions at elements as described above.

Some models can represent variations between computational nodes, or inside computational elements, with a non-linear relationship to help resolve hydraulic conditions in more detail. For example, if an edge is represented with a single elevation, the cross-sectional area over that edge is simply the WSE minus the edge elevation, multiplied by the edge length. However, if that cross-sectional area is represented instead by a nonlinear curve of WSE versus area, the conveyance area of the curve can begin at a lower elevation. At some point, the cross-sectional area over the edge for both approaches converge to the same value.

This capability is described as "sub-grid scale resolution." Without sub-grid scale representation, an element is either wet or dry. With sub-grid scale resolution, the model can represent partial conveyance through an element or over an edge representing partial inundation. Sub-grid scale models compute this non-linear relationship by incorporating additional terrain information along the edge or over the element.

Use of sub-grid scale resolution should be considered with care. With improper placement of element edges, sub-grid scale resolution can result in overestimation of conveyance or provide conveyance where none should exist. A numerical model only computes a single hydraulic solution for each element; therefore, sub-grid scale resolution is not a substitute for adequate resolution set by the element size, configuration, and orientation for reasonable hydraulic solutions. Both model formulations, with or without sub-grid scale features, can provide detailed hydraulic results with meshes that provide an adequate representation of important hydraulic controls.

Numerical models used for transportation hydraulic analysis or design should include sufficient mesh detail to capture hydraulically-significant physical features (terrain) and to represent changes in hydraulic conditions (flow field resolution). Chapter 5 presents recommended best practices for developing efficient and quality 2D model meshes. Chapter 6 presents model review methods to ensure that the simulation is suitable for the intended analysis or design need.

3.4.2. Vertical Velocity Assumption

As discussed in Section 2.3.1, most 2D models fall into the category of depth-averaged models, which means the model computes velocity in the horizontal plane that represents the depth-averaged flow at that coordinate point (x,y). These models include an inherent assumption that the velocity distribution in the vertical direction is negligible. Hydraulic conditions can exist in a riverine environment that contradict this assumption, such as around a physical structure such as a bridge pier, around sharp bends, or down a steep spillway. In situations where the real-world hydraulics contradict vertical velocity assumptions, 2D model results need careful review because they are "wrong," in the sense they don't represent all the hydraulic variables present in such a complex situation. However, these models can still be "useful" due to the averaging methods applied by the model.

3.5. SOLUTION PROCEDURES

The principal output of a 2D model consists of three values at each computation location; water level, velocity in the x-direction, and velocity in the y-direction. To accomplish this, a model assembles a series of PDEs that represent the conservation of mass and momentum for each computation location. By solving this series of equations, the model computes the approximations for the unknown quantities at the computation locations. The method for accomplishing this is dependent on the approach of each computational model (see Section 3.2).

It is not uncommon for element solutions to then be interpolated or averaged to other locations for visualization purposes. The finite element method uses this averaging methodology because the computation locations correspond to gauss points inside the element rather than the element centers or the nodes. Finite volume and finite difference approaches commonly average values from element centers or edges to nodes. This interpolation introduces rounding and potential averaging errors. Most practitioners accept this error potential in exchange for the ability to visualize the solution as a continuously variable quantity.

3.5.1. Implicit vs. Explicit

It is common practice to refer to a numerical solution scheme as being explicit or implicit. One definition distinguishing the two solution schemes relates to the methodology employed for advancing the solution in time. An explicit scheme directly computes the dependent variables in the next time step in terms of known quantities at the present solution time step. An implicit scheme defines the dependent variables using a coupled set of equations involving values from both the current and the next time steps and employs matrix or iterative solver techniques to obtain the solution. Governing equations for computational fluid dynamics are nonlinear and typically involve a considerable number of unknown variables. Because of these conditions, implicit methods almost always use iterative techniques.

The following example describes how the explicit and implicit methods work. Given the value for a variable V at a point in time $t=n\,dt$ (where dt is the time step size and n is the time step index), the numerical method's goal is to compute the value for V at the next time step, when $t=(n+1)dt$. Alternatively, the numerical method computes the rate of change of V at time t (dV/dt) as Δ and estimates V at the next time step as $V_{n+1} = V_n + dt\Delta$.

With an explicit numerical method, Δ would be evaluated in terms of known quantities at time step n. However, an implicit method would evaluate some of the terms in Δ based on terms of unknown quantities at the new time $n+1$. In this implicit formulation, unknown quantities appear on both the left and the right side of the equation. This is referred to as an implicit definition of the new $n+1$ values. Matrix or iterative solution methods are then used to compute the new quantities.

Explicit schemes use much less CPU time to obtain V_{n+1} but are less stable than implicit schemes. The same example may be used to demonstrate the differences in stability of explicit and implicit methods. Dividing the equation for V_{n+1} by dt gives $V_{n+1}/dt = V_n/dt + \Delta$. In the limit, as dt approaches infinity, the equation becomes $0 = \Delta$. For an explicit method, Δ is based entirely on terms from time step n, so no solution exists. Therefore, there is always a practical limit to the size of the time step for an explicit method. Conversely, for an implicit method, the Δ term includes variables from both time steps, so a solution approximation can always be generated. This does not mean the approximation is good, just that it exists, which illustrates the stability of the implicit method.

Explicit methods are generally less complex to represent, easier to program, can be more easily split for parallelization, and require less computational effort for each solution step. However, explicit methods also require a smaller time step resulting in a larger number of solution steps. Consequently, implicit solution methods can require orders of magnitude fewer solution time steps than explicit methods.

Because implicit methods incorporate adjacent information from the $n+1$ time step in the solution for a given element, the numeric solution can dampen or smooth transient and highly localized hydraulic gradients. If these types of conditions are of significant design interest, then a numerical model using an implicit solver may not adequately capture these effects. However, for typical river engineering and roadway encroachment design applications, or for steady-state simulations that do not consider unsteady flow effects, either type of model can be appropriate. The practitioner should evaluate the hydraulic results to ensure the model simulation adequately provides the needed design insight. This evaluation is typically done by performing a sensitivity analysis or varying mesh resolution, as described in section 5.3.3.

3.5.2. Courant Condition – CFL Limitation

The Courant-Friedrichs-Lewy (CFL) condition (Courant, Friedrichs and Lewy 1928), commonly referred to as the Courant condition, is a necessary condition for convergence when solving certain partial differential equations numerically by the finite-difference method with explicit time integration schemes. For these schemes, the Courant condition requires the time step to be less than a threshold interval described below. If it is not, the solution may become unstable or provide inaccurate or physically unrealistic results.

The Courant number, (a dimensionless quantity) compares the wave celerity (v) (the rate at which information can pass through the solution area) to a representative length (the element size, Δx, commonly computed as the square root of the area of an element) and the time step (Δt). The Courant number is stated as $C = v\,\Delta t/\Delta x$.

Explicit methods require a time step size that limits the advancement of flow through the domain to less than one computational element per time step. This restriction is related to accuracy because most finite difference equations involve quantities only from the neighboring elements. A particle that propagates farther than one element in one time step would then be moving into regions that have no defined influence on the variable. Not only is this physically unrealistic, but it also leads to numerical instability. As a result, for explicit methods to be stable, the Courant number should be less than unity (C<1) in the entire model domain.

Implicit methods, on the other hand, couple all the elements together through an iterative solution that allow the transmission of signals through a mesh. The price for this communication between distantly located elements is a damping or smoothing of the variables introduced by the relaxation needed to solve the coupled equations.

Most numerical models are coded in either an implicit or explicit formulation based on the purpose of the model and the choice of the developer. Some models are coded to allow the user to choose the solution method. Therefore, the user should be aware of what numerical method is employed by the model they are using. This information would be in the specific model's documentation. When the solution varies quickly through time, and this variation is the object of the study, an explicit method produces greater accuracy, and an explicit model should be selected (at the expense of shorter time steps). For projects whose objective is a steady state solution or for sites where variations in flow through time are gradual, an implicit method may be more efficient. Normally the principal requirement is that the user select a small enough time step to achieve numerical convergence with the model being used.

3.6. MODEL SIMPLIFICATIONS

Due to the computational expense related to more complex formulations of the physics represented in 2D models, it is common to see formulations that make simplifying assumptions. Modelers should be aware of the formulations employed by a specific model and ensure that the simplified formulation is suitable for the conditions to be modeled. This section describes some of the most common simplifications employed in available 2D models.

While a simplifying assumption does limit the range of situations a formulation applies to, it does not imply that the simplifying assumption should not be made at all. In some conditions, the terms removed by the simplifying assumption are indeed negligible. In situations where the terms are significant, the simplified formulation should not be applied. Generally, available time and

computing resource constraints do not justify the efficiencies gained by simplifying the formulation. See Section 2.4 for additional model selection considerations.

3.6.1. Dynamic Wave vs. Diffusive Wave

The terms "dynamic wave equation" or "shallow water equation" are often used to describe either the full 2D Navier-Stokes equations or the 1D derived Saint-Venant equations to describe the numerics used by hydraulic models. These equations are numerically challenging to solve but are valid for all channel flow scenarios (in 2D or 1D, respectively).

Diffusive and kinematic wave equations are simplified forms of the dynamic wave equations, both of which have found wide application in engineering practice. Diffusive wave models assume that the inertia terms in the equation of motion are negligible when compared with the pressure, friction, and gravity terms. Therefore, the inertia terms (representing acceleration) are removed from the equation. Kinematic wave models make the further assumption that the pressure terms are negligible as compared with the friction and gravity terms.

Kinematic wave equations are the most simplified. However, they can be successfully applied to modeling overland flow, the movement of slow-rising flood waves, and very steep slope channels with no back-water effects. Kinematic wave equations still apply when gravity forces and friction forces balance each other.

Diffusive wave equations apply to all conditions suitable for the kinematic wave equations and do a better job than kinematic wave of representing the subsidence of a flood wave. These models have also been shown to be applicable for a wider range of bed slopes and wave periods than the kinematic wave model. Diffusive wave equations apply when pressure forces are important in addition to gravity and frictional forces. This implies that the momentum forces are still not important.

Dynamic wave equations apply when both inertial (momentum) and pressure forces are important, and backwater effects are not negligible, such as in the case of channels with mild slopes with downstream control. These forces are important for the majority of transportation-related hydraulic cases and should be carefully taken into consideration when selecting a 2D model and the appropriate application of the SWE.

For simulations of riverine situations; the higher the energy of the system, the greater the need for the dynamic wave equation. Bridge crossings and high velocity instream flows should always use the dynamic wave equation.

3.6.2. Rigid Bed vs. Mobile Bed

Another simplification often made when performing hydraulic computations is the separation of hydraulics from sediment transport. A rigid bed model assumes the terrain used for the hydraulic analysis is not mobile and therefore, does not change during a hydraulic analysis. This greatly simplifies the equations for conservation of momentum. Further, the addition of simulation of a mobile bed increases the variables and number of terms being solved for in conservation of various masses. The entire process is at least an order of magnitude more complex.

Mobile bed models have the ability to allow the terrain to change due to hydraulic forces acting on the channel or floodplain surface. Common approaches separate the computation of sediment transport into a process handled between hydraulic time steps.

The current state of practice in transportation hydraulics focuses primarily on rigid bed analyses, with limited application of mobile bed analysis for more complex cases. However, as computational power continues to improve, mobile bed analyses will become more common (see Section 8.4).

3.7.   BOUNDARY CONDITIONS

Simple experimentation with a numeric model such as varying the control depth at a simulation outflow demonstrates that boundary effects have the potential to propagate into the model domain (potentially many floodplain widths). Because of this influence, the modeler should position the boundary conditions in such a way as to keep them as simple as possible and perform sensitivity analysis (see Section 6.4) to ensure that a reasonable range of boundary condition uncertainty does not significantly affect the hydraulic conditions predicted in the area of interest. Therefore, extending the domain limits farther away from the area of interest may be necessary. Preferably, the boundary conditions can be specified at a location where flow approximates simple 1D flow conditions. If this is not possible, extending the domain to an artificially simple region may be recommended (Alemseged, 2007). For further discussion of boundary condition data requirements and assignments, see Sections 4.3 and 5.4.

CHAPTER 4

DATA AND MODELING REQUIREMENTS

Hydraulic models are digital representations of the real world. Various data sources available to the modeler are used to create a digital representation of a specific project location. The better a model represents the location being analyzed, by using high-quality data, the closer the model results emulate actual hydraulic conditions.

This chapter covers information on data required to create and run a 2D model, including sources for common data types. It also contains sections on the primary data types, including terrain, boundary conditions, roughness parameters, background data, and sediment transport analysis. Unless otherwise indicated (by context or citation to statute or regulations), the "requirements" discussed in this document are principles or best practices necessary to ensure scientific reliability, modeling accuracy, or the fidelity of modeling results. This document does not create any new legal requirements.

4.1. DATA OVERVIEW

The amount, quality, and type of data necessary to create 2D models vary depending on the model application and the feasibility of obtaining the data. The terrain data used to create the model geometry has the most significant impact on the quality of the model and typically takes the most resources to obtain a sufficient representation of a project area. Modelers should use the most accurate data available (see Sections 4.2.3). However, it may not be feasible to obtain the "best data" due to economic or accessibility limitations. Even with the best data, some model assumptions limit the model's ability to represent real-world hydraulics precisely. It is the responsibility of the modeler to understand the limitations of the model in relation to the underlying data so model results can be applied appropriately to the problem being analyzed.

The following data sets are recommended to create the mesh geometry for 2D models adequate for transportation-related hydraulic analysis, see Section 4.2 for more information.

- Topographic data, typically lidar, covering the entire model domain

- Supplemental survey data along roadways of interest and under bridge decks

- Bathymetric data representing the channel within the model domain (more detailed survey near important structures such as bridges is recommended)

- Data from surveys and engineering drawings to represent piers, abutments, bridge decks, culverts, and other structures as needed

It is a misnomer that 2D models need significantly more data than 1D models. If the only data available is what has typically been collected for 1D models (i.e., cross section surveys), this data can be used to create a 2D model as long as the data is adequate for 1D modeling and represents all significant changes in geometry, slope, and roughness. However, the modeler should understand the limitation of the model results in the context of the data used to build a model. On projects involving roadways crossing floodplains, it is common for cross section data to be provided to hydraulic engineers only within the roadway right-of-way (e.g., immediately upstream and downstream of a bridge). Using only this data to evaluate the bridge hydraulics is typically not adequate for 1D or 2D modeling. Reasonable survey limits generally extend roughly two

floodplain widths up or downstream of a bridge crossing. Hydraulic analyses using data limits that are significantly less than this are prone to potential errors associated with misrepresented boundary conditions.

4.1.1.   Types and Sources

The specific model application typically dictates the type and accuracy of data necessary for a successful analysis. Typical data needed for 2D modeling include:

- Terrain data (digital elevations from lidar, photogrammetry, or field surveys)
- Upstream boundary conditions (discharges)
- Downstream boundary conditions (fixed WSE, rating curve, or tidal condition)
- Land use for model domain (hydraulic roughness parameters, Manning's n)
- Hydraulic structure geometry (bridges, culverts, and other inline or diversion structures)
- Aerial imagery, CAD data, and GIS (not required but helpful)
- Application-specific information (for example, soil characteristics for sediment transport studies)

Although all data used for a model is important, the mesh geometry (see Chapter 5), developed from terrain data, has the most significant impact on model results and is the greatest source of uncertainty and error (Figure 4.1). Recognizing the relative importance of each data type assists modelers in directing how resources, such as time and budget, are spent developing each type of data, and related model components.
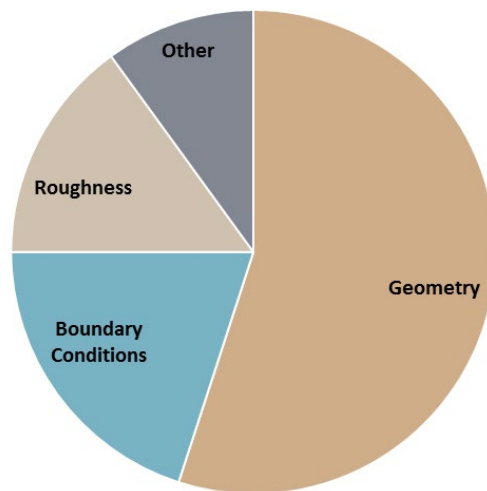
Figure 4.1. Approximate importance of data type to model results.

Terrain data is used to create the mesh geometry (see Section 4.2.), which includes elevations above water (topography) and below water (bathymetry). Multiple terrain data sources are typically required to create a digital representation of the entire region to be analyzed.

Boundary conditions (see Section 4.3.) typically include the discharge of water entering the model domain and the hydraulic control (WSE) at locations where water is exiting the model domain. A 100-yr discharge and a downstream WSE are example boundary conditions. In tidally influenced areas, the downstream boundary condition may consist of a WSE time series.

The land use over the model domain is needed to define the hydraulic roughness for every mesh element. The hydraulic roughness parameters (see Section 4.5.), typically Manning's n values, are developed from standard resources that provide estimates for various land use types, which are determined from aerial imagery, site photos, sediment data, or proposed plans.

Hydraulic structure geometry (see Section 4.4 & 5.5), such as bridges, culverts, weirs, and gates can be represented in some 2D models, and are necessary when such structures are hydraulically significant to the analysis.

Several data sources that often have valuable information for developing 2D models are accessible online. Table 4.1 provides a sample list of national data sources.

Table 4.1. Data sources for 2D hydraulic models

Source	Data Types			
	Terrain	Boundary Conditions	Roughness	Imagery and GIS
United States Interagency Elevation Inventory	X			
The National Map	X	X		X
USGS Earth Explorer	X			X
USDA Data Gateway	X			X
NOAA Bathymetry	X			
USGS Topographic Maps				X
USGS Hazards Data Distribution System				X
NGS Data Explorer	X			
USGS Water Information System		X		
NOAA River Observations		X		
USGS Streamstats		X		
NOAA Hydrometeorological Design Studies Center		X		
USGS National Streamflow Statistics Program		X		
FEMA Flood Map Service Center		X		X
Manning's n Values			X	
USGS Guide for Selecting Manning's Roughness Coefficients			X	
USGS Calibrated channel Manning's n Values			X	
Note: Click on each source title to link to the website				

Similar data sources exist for several locations at state, county, regional, and municipal levels. Modelers should become familiar with available local geospatial data resources. Online data sources will likely not provide all data required to create and run a 2D model for detailed analyses, but they may be sufficient for planning purposes in many cases. Examples of additional data typically needed for detailed hydraulic analyses include:

- More detailed topography
- Bathymetric data of the channel (cross sections or more complete mapping of the channel)
- Bridge/culvert as-built plans
- Bridge/culvert survey (including channel topography below the bridge deck)
- Weir/diversion structure survey
- Hydrologic data

4.1.2. Coordinate Systems and Datums

Data used for 2D models should be referenced to a horizontal and vertical datum. A datum is a fixed starting point, or "0", for a unit of measure. A horizontal datum is often referred to as a coordinate system or projection. Hydraulic models can be created and run successfully in any coordinate system or vertical datum. The modeler needs to be aware of the horizontal and vertical datum used for the model and assure all data sources used in the modeling process are consistent. It is helpful, although not required, to have project data that is referenced to a known coordinate system, to best use the available online data sources. Some states and communities work in a local coordinate system. Local systems can be referenced to other common coordinate systems as long as the conversion to these systems is known or defined.

Horizontal Datum. All coordinate systems are based on a spherical horizontal datum such as the World Geodetic System 1984 (WGS 84) and the North American Datum 1983 (NAD 83). Two common types of coordinate systems are geographic and projected. Geographic coordinate systems (GCS) use a sphere to define locations on the earth. The shape of the earth is typically modeled as an ellipsoid. Latitude and longitude reference a point on the ellipsoid. Global positioning system (GPS) surveys are based on a GCS. Mapping applications such as Google Earth use a GCS (Figure 4.2).
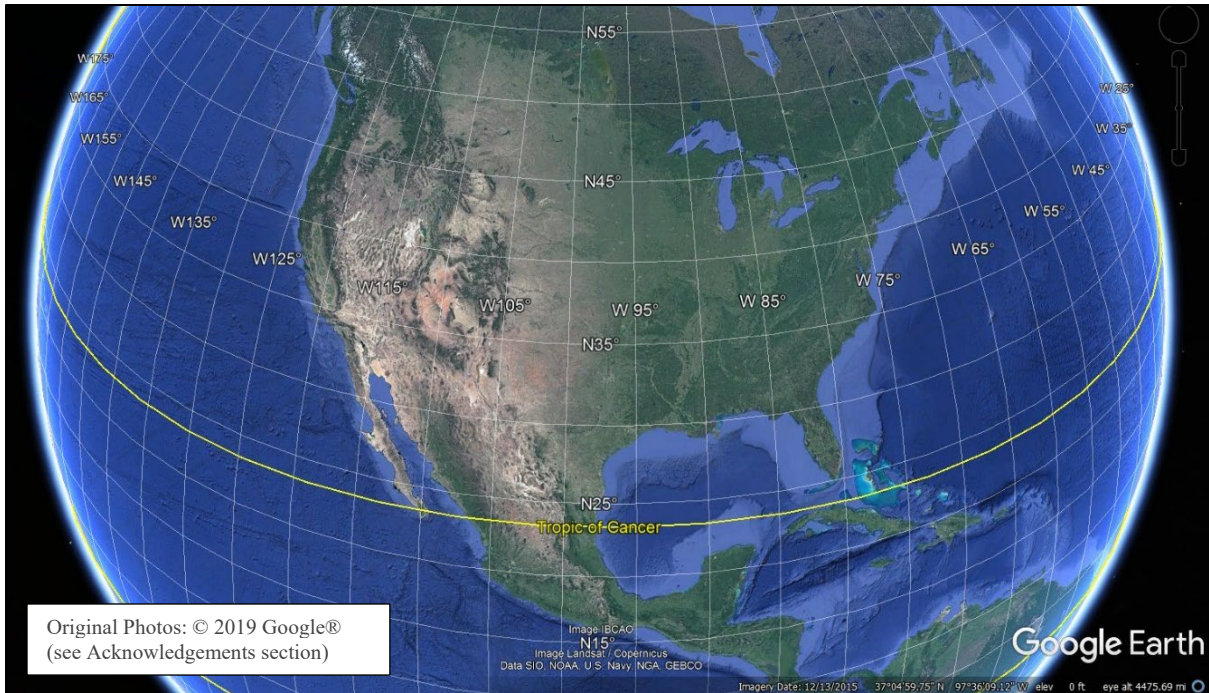
Figure 4.2. Global coordinate system latitude and longitude displayed in Google Earth.

Projected coordinate systems (PCS) represent a sphere as a flat surface. A PCS includes projection parameters that customize the projection to a specific location and system of units. Typical PCS include Universal Transverse Mercator (UTM) (Figure 4.3), State Plane Zones (SPZ) (Figure 4.4), and local projections.

Survey data can be in "grid" or "ground" coordinates. Both coordinates can be used for 2D models, but it is important for users to understand what coordinate type the data is in, so separate data sets can be used together. Grid coordinates include common SPZ and UTM projections previously mentioned. The term "grid" refers to representing locations from the curved surface of the earth on a flat grid. Representing a curved surface introduces an error related to the true length of a line. Ground coordinates refer to survey data that correctly represent the distances as would be measured along the curved surface of the earth. A correction factor can be used to convert between the two coordinates systems. Many state Department of Transportation (DOT) survey divisions have developed projections files (.prj) that specify the coordinate system and grid/ground conversion factors for the entire state or specific regions.

When using the US standard system of units for PCS, it is important to determine if the US survey feet or international feet are required. The difference is how a foot is calculated and are defined as:

- International foot = 0.3048 meters
- US survey foot = 1200/3937 meters

The difference is negligible for vertical units and small measurements but is significant when working with horizontal coordinate systems such as a state plane system. Because the coordinates for state plane zones are in the order of millions, the small difference between the international and US foot may lead to a horizontal uncertainty of up to 10-15 feet. Modelers should ensure the consistency of coordinate systems for all data sets.
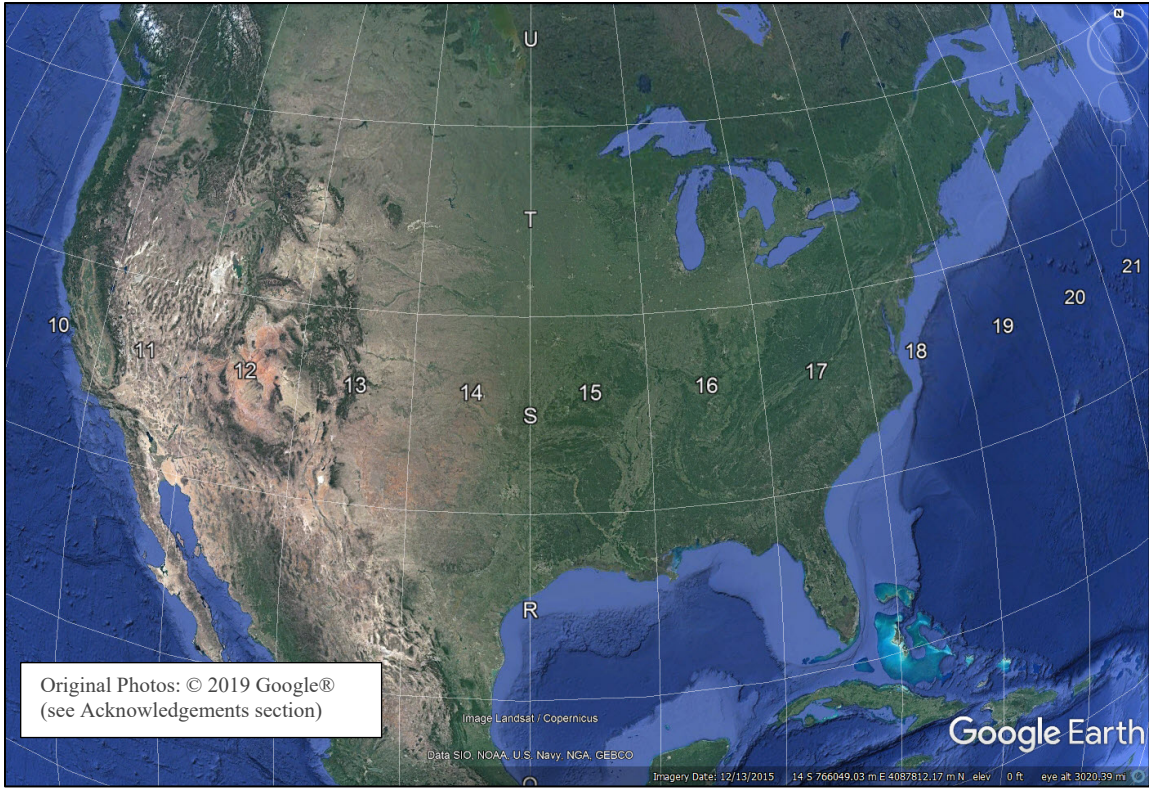
Figure 4.3. UTM zones displayed in Google Earth.
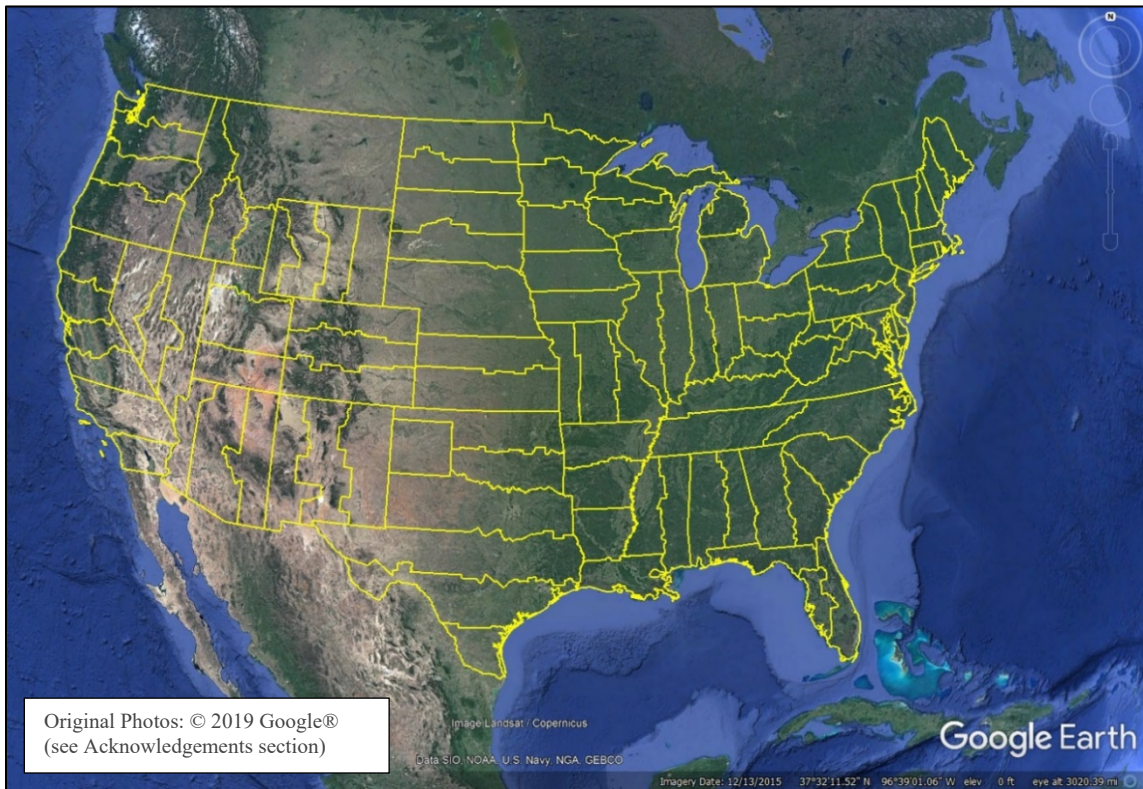
Figure 4.4. State plane zones displayed in Google Earth.

<u>Vertical Datum</u>. Any data that includes elevation is referenced to a vertical datum. Common vertical datums include National Geodetic Vertical Datum of 1929 (NGVD 29), the North American Vertical Datum of 1988 (NAVD 88), and local/custom vertical datums. Using incorrect, or multiple, vertical datums on a project can have substantial impacts on the accuracy of the model and the resulting design. It is therefore imperative that modelers verify the vertical datum of all elevation data sources and ensure that all data are converted into a common project vertical datum correctly.

Data sets with differing vertical datums have to be adjusted so that all data is referenced to the same vertical datum. Differences between NGVD 29 and NAVD 88 vary depending on geographic location and can be determined using the National Geodetic Survey (NGS) online tool Vertcon. Data in local vertical datums, such as from historic bridge plans, need to be converted to the project datum. The conversion can be accomplished by obtaining multiple survey points in the project vertical datum at locations where the local elevation is known so that the appropriate vertical adjustment can be determined. For more information about conversions, see FEMA's "Guidance for Flood Risk Analysis and Mapping, Vertical Datum Conversion" (FEMA 2014).

Horizontal and vertical datums have been, and continue to be, adjusted due to improved measurement capabilities and errors recognized in old datums (see Figure 4.5). The National Geodetic Survey (NGS) is in the process of creating an updated, more accurate, horizontal, and vertical datum for the US. The new datums, to be published in 2022, will replace NAD 83 and NAVD 88 as the latest and most accurate datums for the US. The NGS, in cooperation with multiple state and federal agencies, produced a document on the effects of the updated datums on floodplain mapping (NGS 2011).
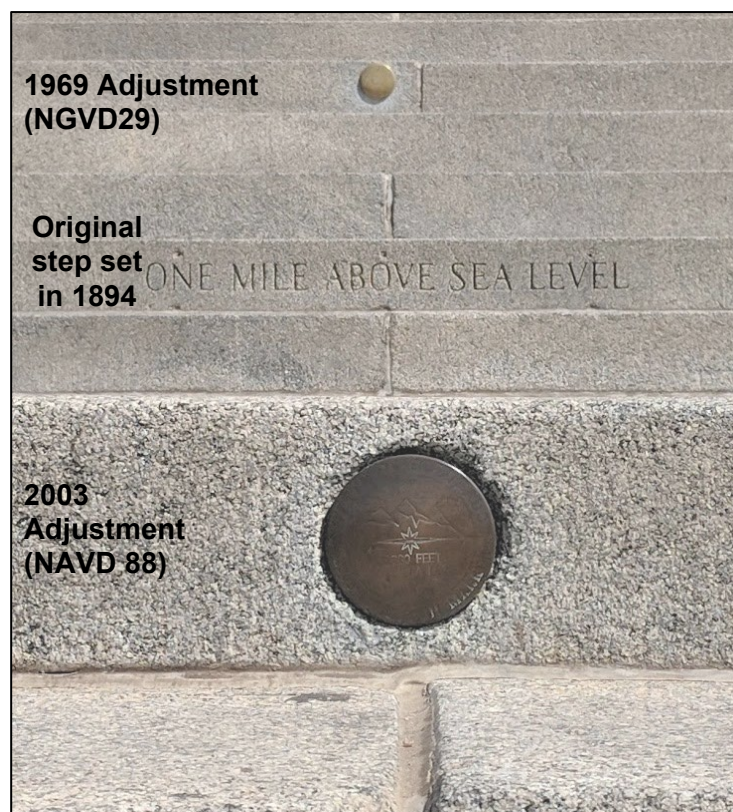
Figure 4.5. Colorado State Capital steps showing multiple adjustments made to the one mile above sea level location.

4.1.3. Metadata

During the creation of a 2D model, several data sets are typically downloaded, collected, trimmed, adjusted, and merged. For the modelers, reviewers, and future model users, it is good practice to include metadata for all data sources. Metadata is information about specific data sets. Data sources should be documented in a report; however, it is also recommended that metadata be included in the model files so important information is contained within the model. Methods for documenting metadata vary depending on the software package being used. At a minimum, it is recommended that the types of metadata shown in Table 4.2 be included with the model.

Table 4.2. Recommended metadata to include with model.

Data Type	Metadata
Terrain (may include multiple sources/surveys)	Source
	Type
	Date
	Coordinate system
	Vertical datum
	Stated accuracy
Boundary Conditions	Source
	Confidence in accuracy
Roughness Parameters	Source (i.e., imagery)
	Values
	Assumptions
Hydraulic Structures	Source of data
	Coordinate system
	Vertical datum
Background, GIS, other	Source
	Coordinate system
	Vertical datum

4.2. TERRAIN DATA

In 2009 The National Research Council published a study titled "Mapping the Zone," with the purpose of improving flood map accuracy (NRC 2009). The study was requested by managers of the Federal Emergency Management Agency (FEMA) Risk Analysis Division, the National Oceanic and Atmospheric Administration (NOAA), NGS, and the Coast Survey Development Laboratory. The Committee on FEMA Flood Maps was established to examine the factors that affect flood map accuracy and identify ways to improve flood mapping among other tasks. The first overarching finding was that "Topographic data are the most important factor in determining WSE, base flood elevation, and the extent of flooding and, thus, the accuracy of flood maps in riverine areas."

While this study focused on floodplain mapping, which is primarily conducted with 1D modeling, this finding is equally relevant for 2D model applications. The terrain data and the geometry developed for the model have the most significant impact on the accuracy of the model results. It is therefore crucial for modelers to understand the collection methods, their accuracy, data formats, and typical processes and workflows related to the most critical data type related to 2D modeling. This section provides information on these topics specific to hydraulic modeling. Several existing resources from government agencies, such as the USGS, NOAA, and other organizations, such as the American Society for Photogrammetry and Remote Sensing (ASPRS), provide more in-depth information related to most of the topics covered. A recommended resource is the recent "Digital Elevation Model Technologies and Applications: The DEM User's Manual, 3rd Edition" (ASPRS, 2018).

4.2.1. Hydraulic Controls

Terrain data is the most important data set for 2D modeling because the terrain has the most influence on the movement of water for open channel flow, compared to other data types. The portions of terrain having an impact on hydraulics are referred to as hydraulic controls (see Figure 4.6 and Figure 4.7). Typical hydraulic controls include:

- River channel geometry
- Grade control within the channel (riffle, weir, dam, diversion)
- Grade transition from channel to floodplain
- Flow paths within the floodplain
- High ground within the floodplain
- Natural constrictions within the channel or floodplain
- Roadway embankments
- Bridges, culverts, and other hydraulic structures (not always represented by mesh geometry)
- Surface roughness (represented by Manning's n values) is also a hydraulic control and is discussed in Section 4.5

Before developing a 2D model, modelers should verify that the final terrain surface represents hydraulic controls as they relate to the purpose of the model. During the development of the model, the mesh geometry should be created to represent the hydraulic controls in the terrain data (see Section 5.3.3). If the hydraulic controls are not represented with sufficient accuracy in the terrain and mesh geometry, the model is not able to represent the hydraulics of the project location. Hydraulic controls and terrain data requirements may vary depending on the model application and discharges analyzed. Terrain features controlling hydraulics at low discharges may not have any hydraulic impact for high discharges. Similarly, features controlling hydraulics for larger discharges may have no impact for smaller discharges.

A stream rehabilitation study with a small channel and floodplain may necessitate more detailed representation of small features because they may impact hydraulics at a scale that is important to the rehabilitation plan. In contrast, a floodplain study on a large river system with a wide floodplain is unlikely to need as much small-scale detail of the channel and minor terrain features in the floodplain.
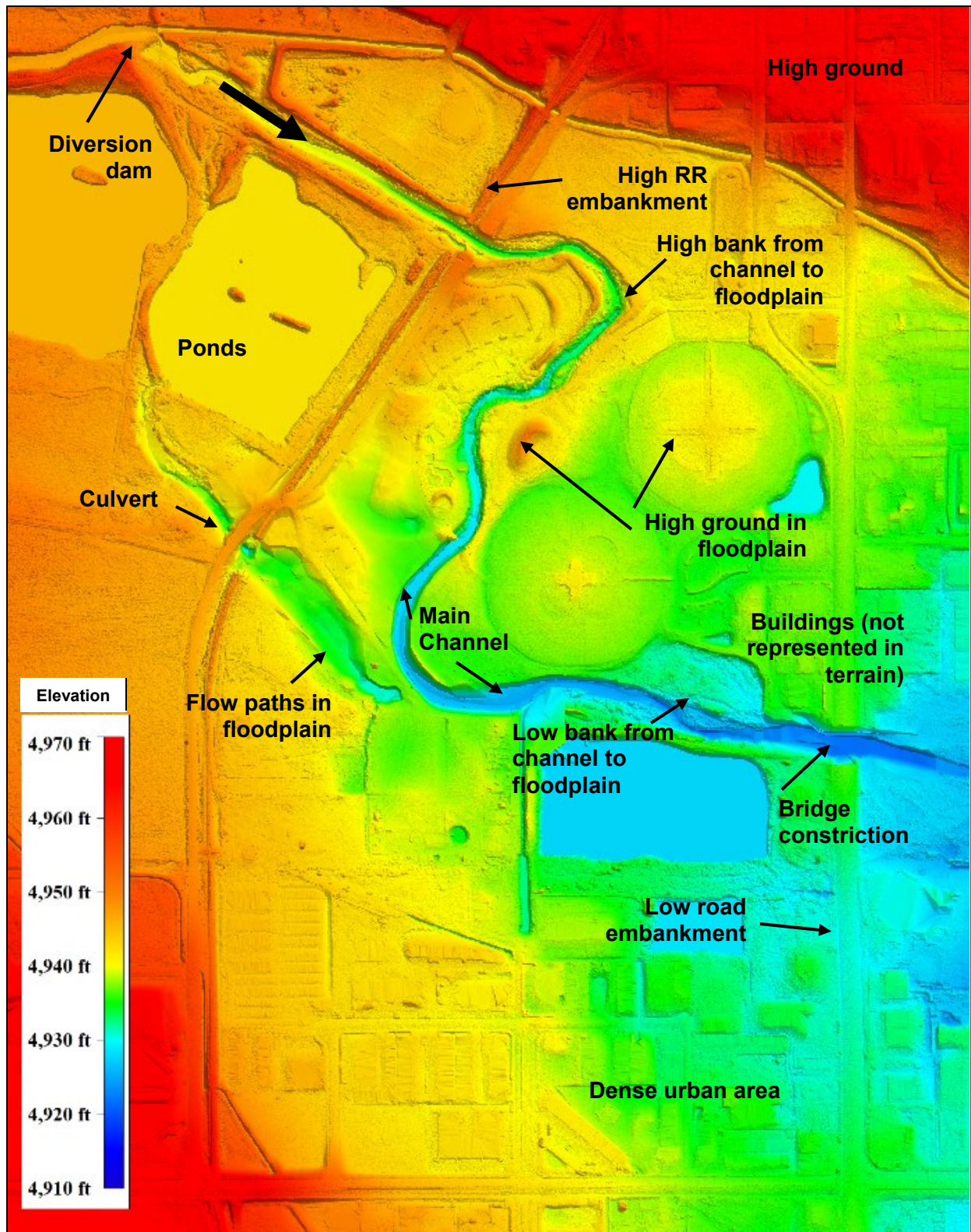
Figure 4.6. Hydraulic control examples.
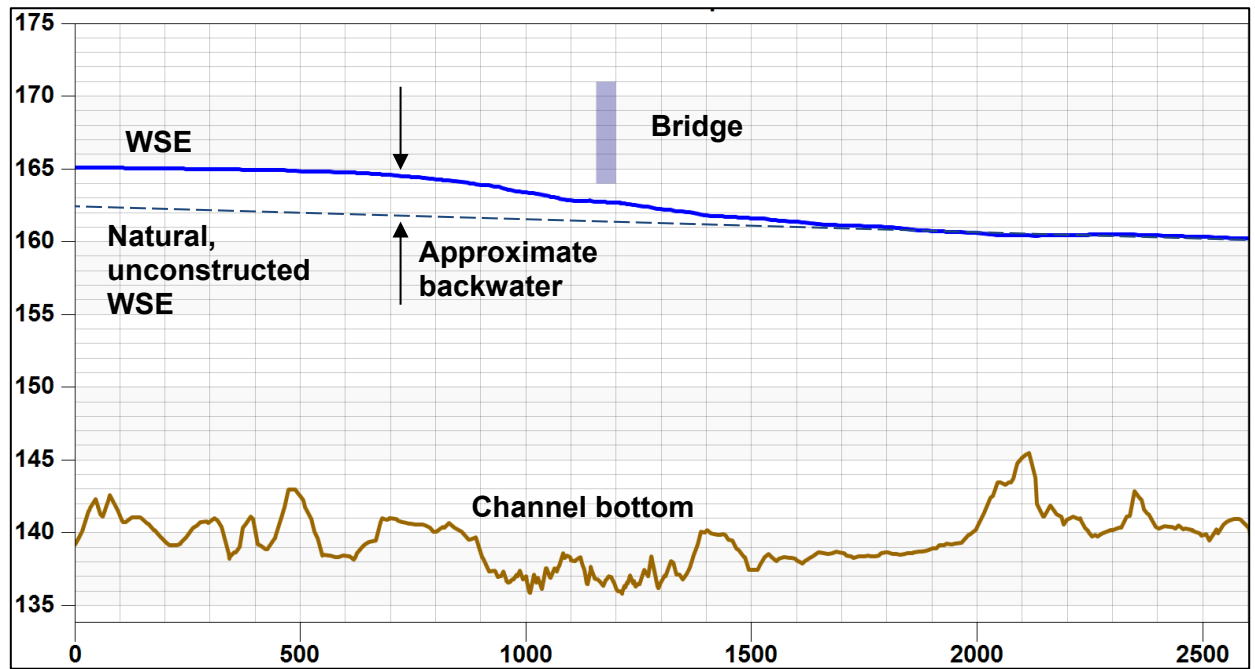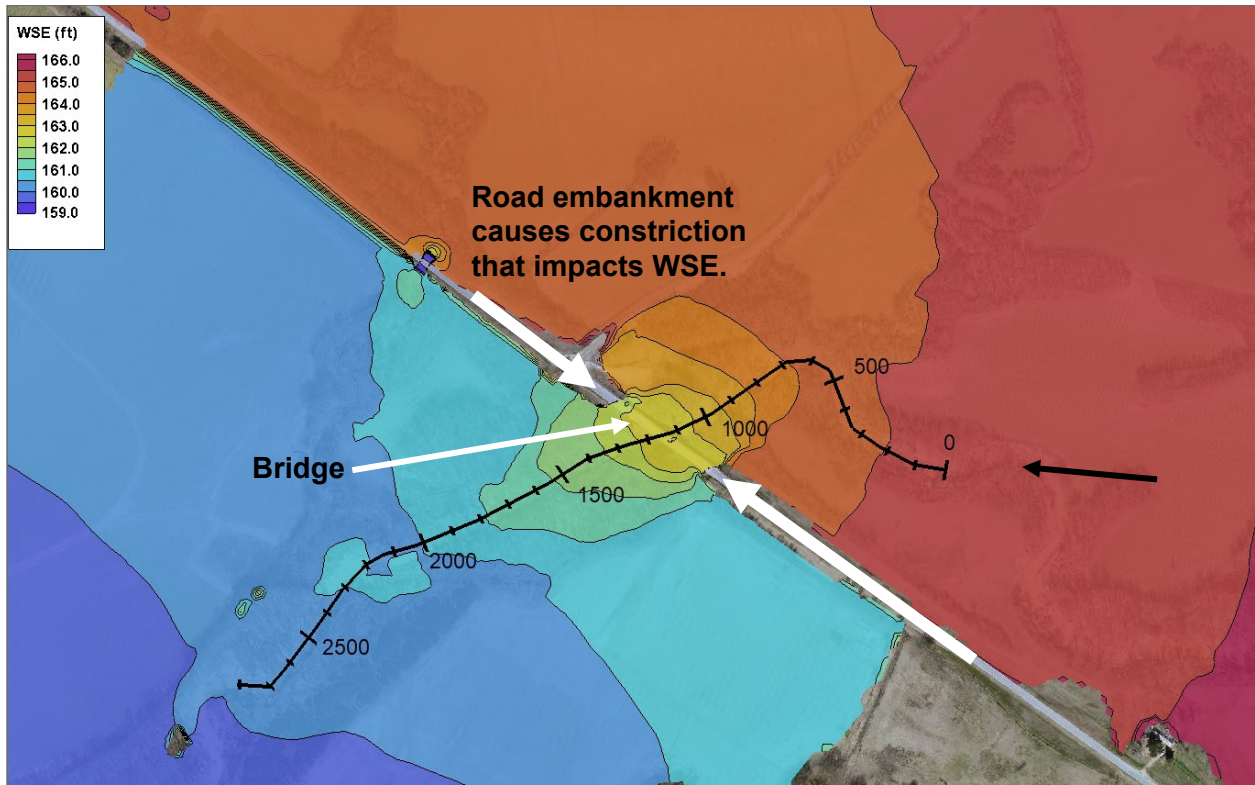
Figure 4.7. Plan and profile showing hydraulic impact of roadway embankment.

To appropriately assess the sufficiency of the terrain data, modelers need to understand the application of the model, the range of discharges to be analyzed, the area of interest, and the limitations of terrain data collection methods used. If additional data collection is not feasible, then the modeler should understand the potential limitations of the model and communicate those limitations to stakeholders. While assessing the sufficiency of the terrain data, the mesh resolution of the model should also be considered (see Section 5.3).

4.2.2. Collection Methods

Modelers may download online data, provide direction to surveyors or staff for the collection of data to be used in a model (see Appendix B), or may combine multiple data sets from various sources into one seamless terrain surface. Understanding the basic concepts and limitations of various collection methods used to obtain terrain data are important for these tasks.

A variety of collection methods exist for developing topographic and bathymetric data. Each method has benefits for various data requirements as well as limitations that end-users should understand. A list of conventional data collection methods, with ideal uses and limitations, is provided in Table 4.3. A brief overview of each method is provided below.

Table 4.3. Common terrain data collection methods.

Type	Method		Ideal Use	Limitation
Topographic	Aerial Lidar	Traditional	wide area (county, municipality, watershed, etc.) and long corridors	cannot penetrate some dense vegetation types, cannot penetrate water
		Geiger mode		
		Flash		
		Photon		cannot penetrate clouds, cannot penetrate some dense vegetation types
		UAS		dense vegetation
	Photogrammetry	Traditional	sites and narrow corridors	lower surface definition and limited by dense vegetation
		UAS	small areas and corridors	cannot provide ground data under vegetation
	Ground Survey	GPS	detailed site data, structure geometry	limited to small areas, requires some view of sky for GPS signal
		Total Station		limited to small areas, requires line of sight to object being surveyed
		Scan (lidar)		
Bathymetric	Lidar	Bathymetric	coastal waters, inland waterways	limited to shallow, non-turbulent water with low turbidity
	Sonar	Single-beam	small waterways	cannot survey in turbulent/aerated water, limited to locations accessible by boat
		Multi-beam	medium-large waterways	
		Swath		
	Ground Survey	GPS	small waterways	locations accessible by wading, limited to small areas
		Total Station		

<u>Aerial Lidar.</u> Lidar is currently the most common type of aerial survey method and is ideal for capturing high-resolution topographic data of large areas and corridors. Lidar data is presently available for over half of the continental US (see Figure 4.8). The goal of the USGS 3D Elevation Program's (3DEP) is to provide national lidar coverage of the entire US by 2023.
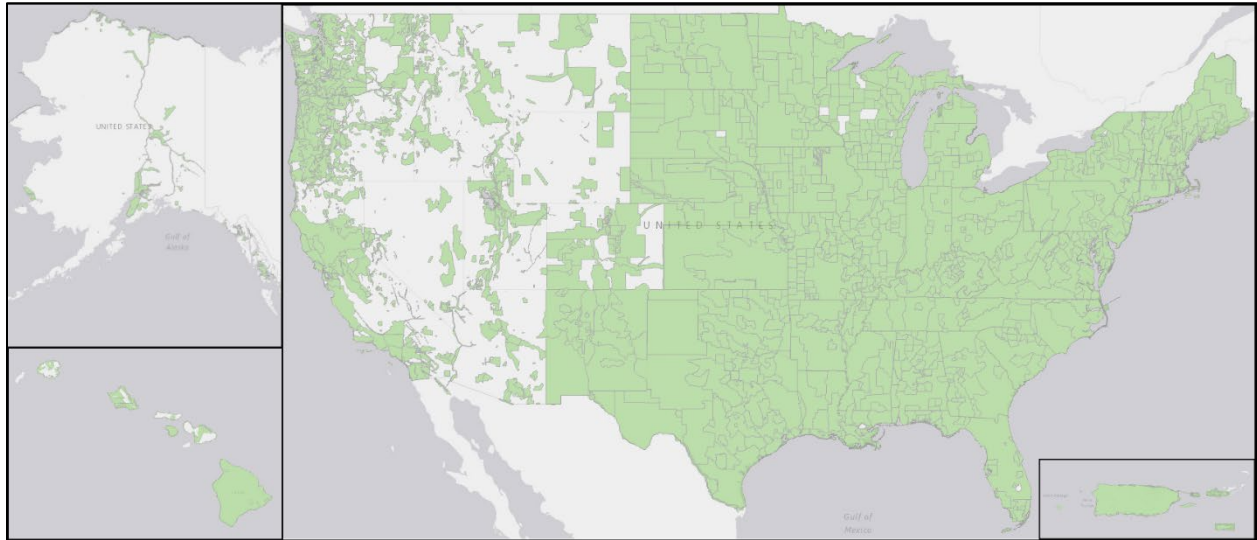
Figure 4.8. Approximate lidar availability in the US (green areas), as of August 2019, from US Interagency Elevation Inventory.

Traditional lidar involves a laser light pulse emitted and detected by sensors that measure the time, and therefore distance, it takes for the beam of light to reflect off an object (see Figure 4.9). The distance measurement, along with the precise position and angle of the sensor, allows the lidar system to determine the three-dimensional coordinates of a location on the ground.

Newer lidar technologies, including Geiger-mode, single-photon, and flash lidar, are increasing in use and have the potential to provide faster collection times and surface definition at comparable accuracies. From a user perspective, the end products are similar. Each collection method has strengths and weaknesses, with the specific mapping application dictating what collection method is appropriate.

Aerial lidar systems used for mapping large areas are housed in fixed-wing aircraft. Lidar sensors can also be attached to helicopters, automobiles, boats, tripods, backpacks, and handheld devices. While the application, point density, and accuracy for these methods vary, the underlying technology is similar.
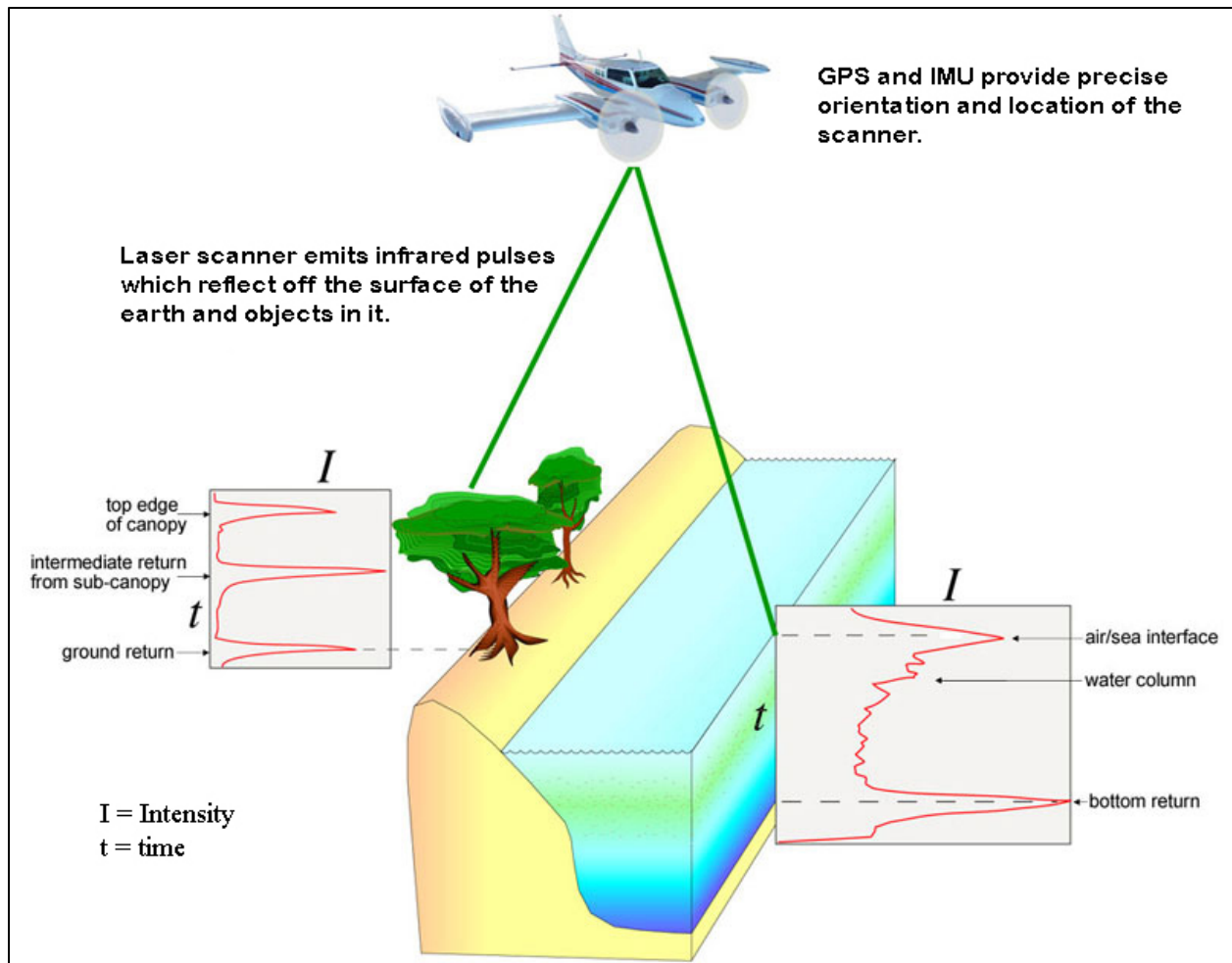
Figure 4.9. Aerial lidar survey schematic including traditional and bathymetric capabilities.

Lidar systems rapidly measure the earth's surface at sampling rates of 50,000 to over 1,000,000 points per second. Aerial lidar surveys produce a dense network of highly accurate georeferenced points referred to as a point cloud. The points can be triangulated together to create a three-dimensional representation of the earth's surface. Lidar is controlled using airborne GPS and precise ground control coordinates established at a higher order than the stated accuracy objective of the final topographic surface model.

The ground footprint of a laser emitted from aerial lidar sensors can vary in size but is typically the size of a manhole cover. The sensor can distinguish multiple returns from objects the laser contacts (Figure 4.9). For instance, the first return might be the top of a tree followed by multiple lower tree branches, understory vegetation, and finally, the ground surface. If the light is visible from the ground, then lidar sensors can feasibly measure that location. Lidar sensors can work at night, but one limitation is that the laser cannot penetrate cloud cover.

Even though lidar has the ability to "see through vegetation," it is best practice to collect lidar data during "leaf-off" conditions in highly vegetated areas such as riparian corridors. Even with a high number of sampling rates, and ideal conditions, lidar cannot map the ground surface through some types of vegetation. For instance, certain crops, coniferous trees, and dense, low vegetation can prevent the laser from reaching the ground surface.

The first product of a lidar survey is the raw point cloud (see Figure 4.10). This data set contains all data returns recorded by the sensor and includes trees, houses, cars, people, power lines, and sometimes can include returns off birds or through storm grates. While the unprocessed raw point cloud can be insightful and applicable for some uses, it should not be used for hydraulic analysis.
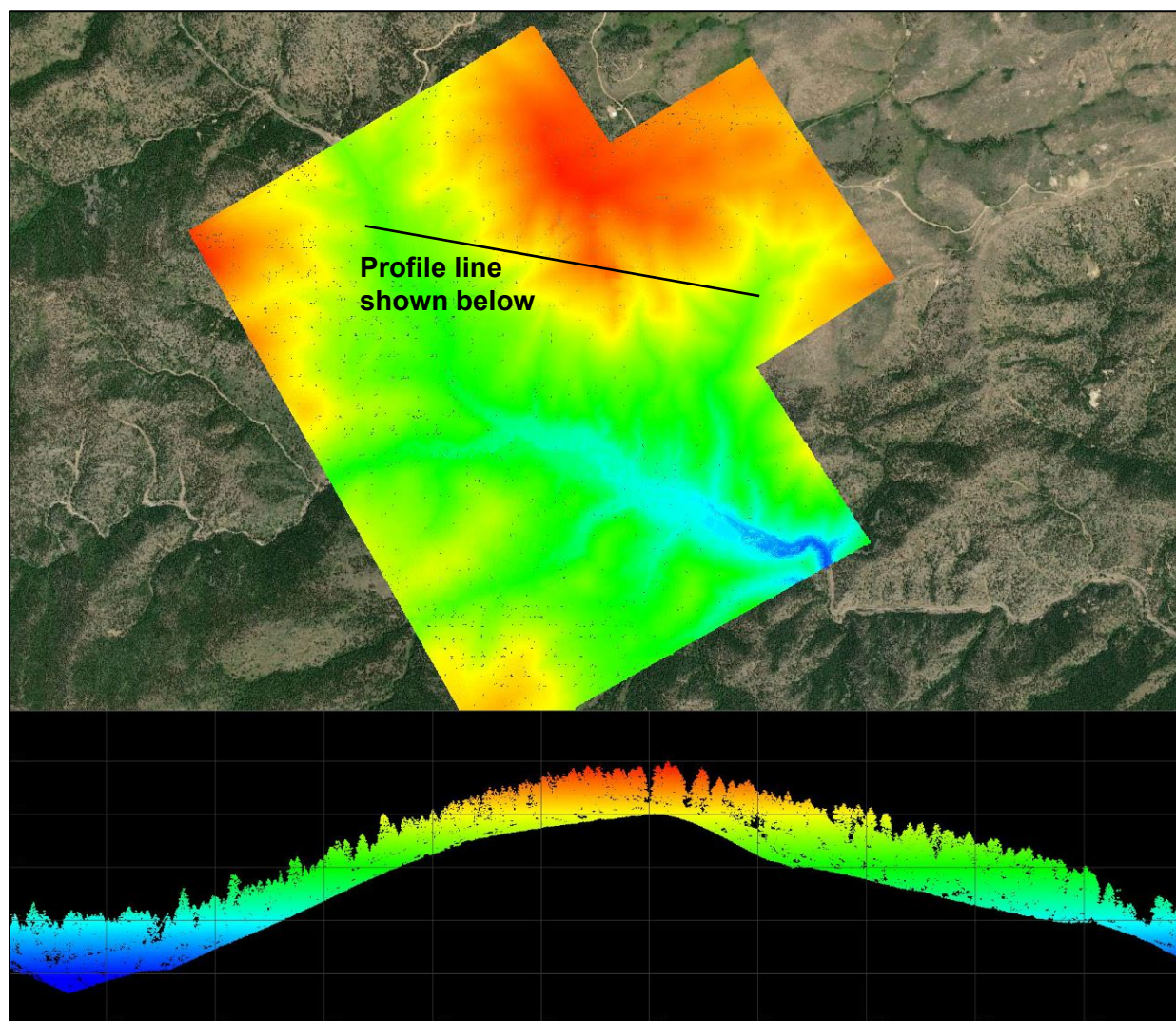
Figure 4.10. Typical lidar raw point cloud, plan, and profile.

The raw point cloud should be processed and classified by a geospatial professional experienced with lidar data. Lidar processing allows the tagging of individual points based on what each point represents. For example, a paved surface, roof, top of vegetation canopy, top of understory vegetation, and ground. The ground points are of interest for hydraulic modeling since they represent the ground surface with all other points "stripped away." A common deliverable for lidar data is the "bare earth" data set that only contains the ground points (see Figure 4.11).

In addition to processing lidar data to bare earth, one additional refinement is the development of hydro breaklines. Hydro breaklines represent the edge of the water (at the time of the survey) along water bodies and rivers. While not always required during lidar data processing, they are beneficial to hydraulic modeling and should be requested if possible. The USGS requires that breaklines be compiled for rivers and streams greater than 100 feet wide and ponded water

greater than 2 acres for their 3DEP lidar program. In some instance, it may be appropriate to request breaklines for smaller and narrower water bodies.

Figure 4.11. Raw vs. bare earth triangulated lidar data.

The lidar return of a water surface can introduce errors due to how the laser reflects off the water. The only reliable lidar point on water is one that is looking straight down (the nadir of the laser). Because of this, all the points on water are removed to produce the bare earth surface. When all the ground points are triangulated together, incorrect triangles cross water bodies and river channels. Hydro breaklines are incorporated into the triangulation to correct this (see Figure 4.12). This process is generally referred to as "Hydro-flattening."

Processed lidar data that meets a desired specification still requires a review by the hydraulic modeler to determine if it correctly represents the hydraulic controls within the area to be modeled. A final lidar bare earth data set typically includes locations that need additional data to represent a project site sufficient for hydraulic analysis. Some locations may not be correctly represented due to limitations in the technology or errors during processing. Ground survey data should be used to verify the accuracy of the lidar data. See Section 4.2.6 for a list of common issues encountered when working with terrain data.
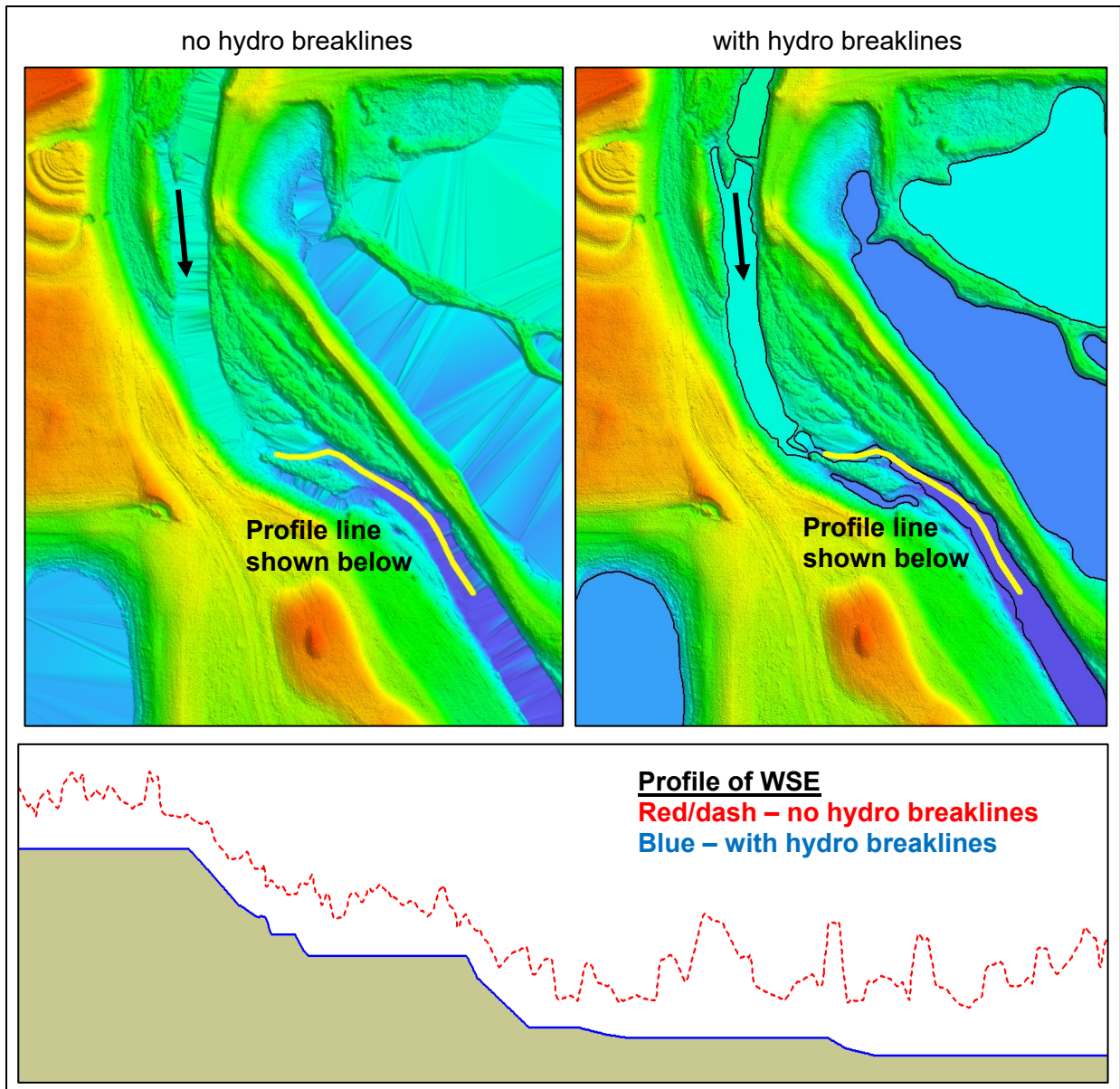
Figure 4.12. Bare earth data set without and with hydro-flattening-breaklines and comparison profile.

<u>Aerial Photogrammetry.</u> Photogrammetry is the science of making measurements from photographs. Similar to lidar, photogrammetry can be obtained from cameras on multiple platforms, including handheld, vehicle-mounted, and many types of aircraft. An example of typical processed photogrammetric data is shown in Figure 4.13.

4.17
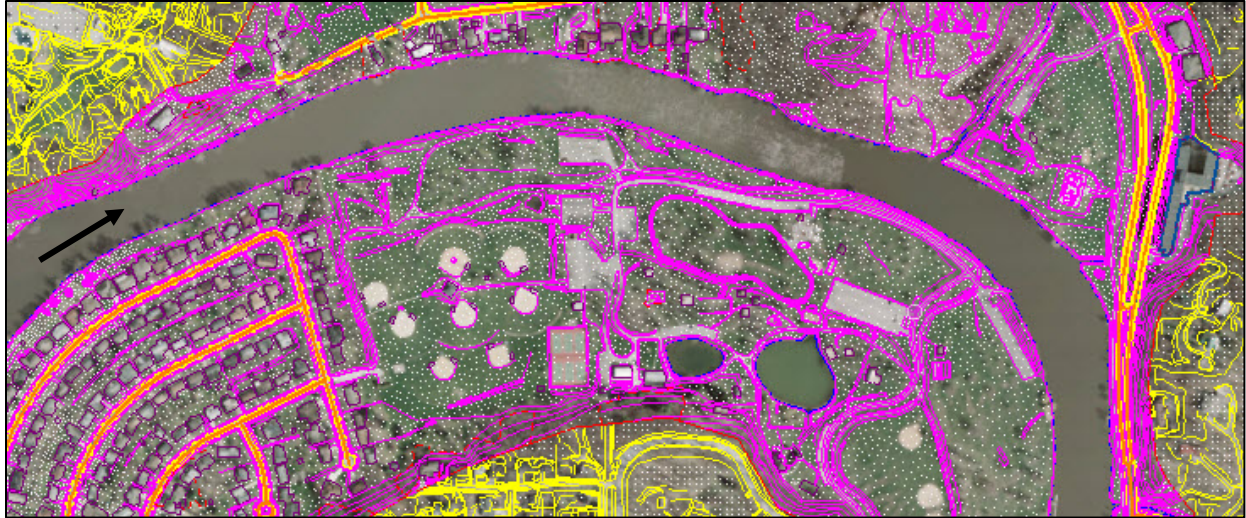
Figure 4.13. Traditional photogrammetric data.

With aerial photogrammetry, 3D coordinates of points on the ground can be determined by knowing the location and metrics of the camera, known and well-defined surveyed control points on the ground, and observing a point from multiple camera angles (Figure 4.14). Traditional aerial photogrammetry involves the use of specialized cameras mounted in a fixed-wing aircraft. Flight lines are planned to produce several images of an area to be mapped with each image overlapping adjacent images. This process yields 3D image-pairs, or models, that enable extraction of information in-stereo, commonly referred to as "3D stereo-compilation".
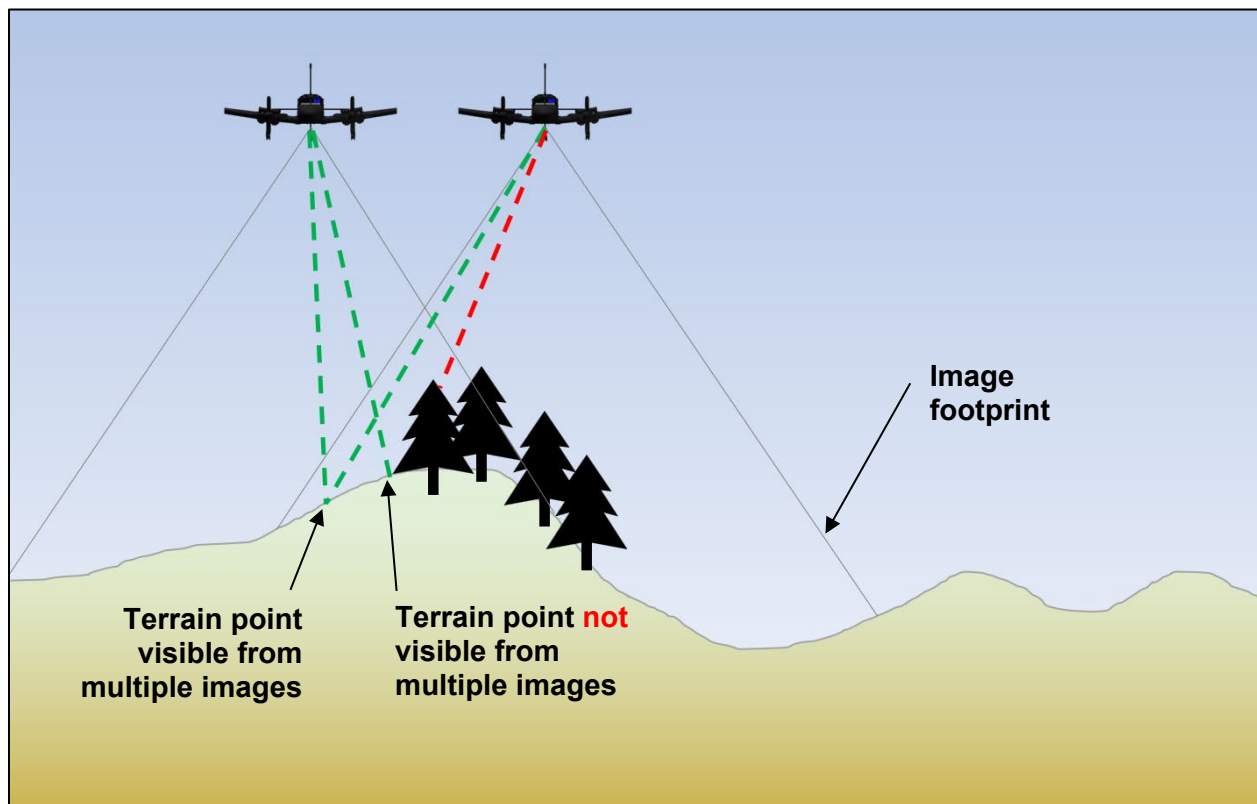
Figure 4.14. Photogrammetric process of measuring 3D points from overlapping imagery.

The collection process is relatively similar, no matter what type of aircraft and camera is used to collect the data. Photogrammetric mapping for a surface model can be labor-intensive. This is one of the reasons that lidar is the predominant aerial remote sensing method used today for large-area surface mapping. Three-dimensional stereo-compilation remains the principal method of obtaining planimetrics (transportation infrastructure, dwellings, impervious surface, utilities, and other structures).

Photogrammetric data can be collected using unmanned aerial systems (UAS). These systems can be used to perform manual terrain mapping through traditional 3D stereo-compilation or semi-automated generation of an image point cloud, using a process called "Dense Image Matching" (DIM). DIM is similar to first return lidar point clouds and is often processed and edited in a similar way (Figure 4.15). Because UAS fly at lower altitudes, the resulting point cloud provides a significantly denser data set than traditional photogrammetric methods.
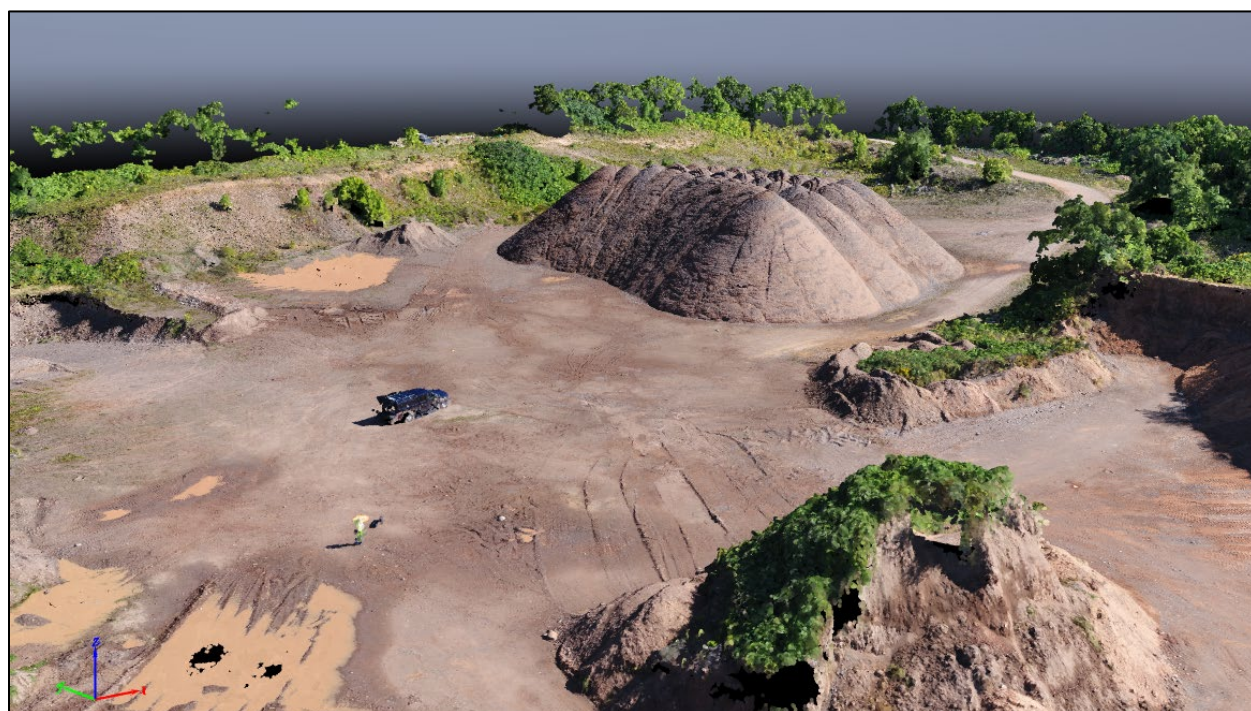
Figure 4.15. Colorized point cloud of photogrammetric data collected by UAS.

It is essential for users of photogrammetric data to understand one of the primary limitations of the collection and processing procedure. For a point to be mapped, it has to be visible from two different camera positions (see Figure 4.14). Because of this limitation, photogrammetric mapping may not be the best solution for locations with dense vegetation, such as vegetated riparian corridors or obtaining data below the water surface. Additionally, photogrammetric imagery should be collected under specific environmental conditions, and appropriate sun angles to limit shadows.

Ground Survey. Surveys conducted with ground-based, or terrestrial, instruments have the potential to be the most accurate data collection method. A significant part of the error associated with aerial survey methods comes from determining the location of the survey instrument in the air. When using ground-based instruments, it is possible to locate the survey instrument with a high degree of accuracy. Typical ground survey methods include:

- GPS
- Total Station
- Terrestrial Scanner (lidar)

Ground-based surveys are typically used to collect detailed data in locations where aerial methods did not provide sufficient coverage, in cases where better data is necessary, or where other survey methods are not feasible. Example locations typically collected by ground survey for 2D modeling include the following.

- Under bridge decks
- Bridge and pier geometry
- Culvert geometry
- Other hydraulic structure geometry data
- Roadway corridors of interest to a hydraulic model
- Bathymetric data in wadeable streams

While each point from ground-based surveys can be accurate to less than 1 cm, the combined surface is typically not that accurate due to the limited number of points a surveyor can feasibly collect. Because of the limitations of the collection methods, it is not feasible to collect data over large areas using ground-based survey methods. Figure 4.16 shows a typical ground-based survey collected with GPS or a total station. Survey data collected with terrestrial lidar results in a point cloud similar to UAS data or dense aerial lidar.
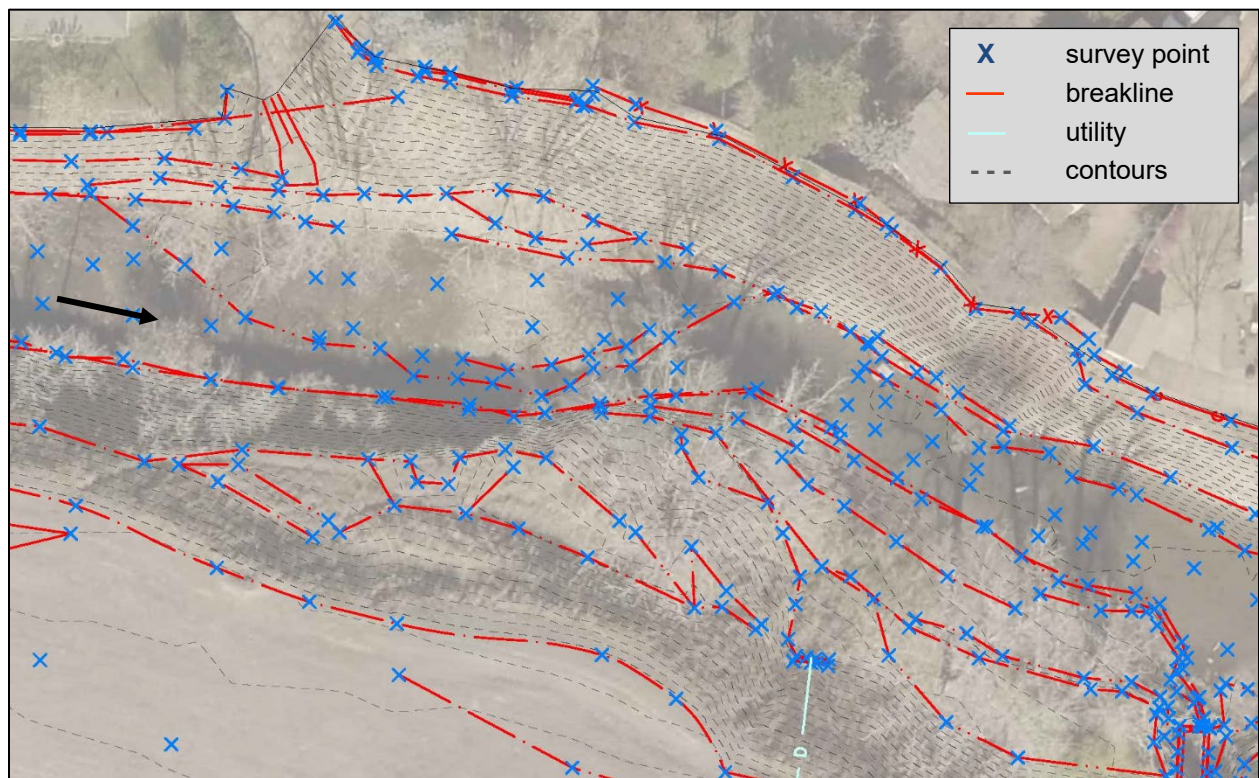
Figure 4.16. Ground survey data example.

Bathymetric Lidar: Bathymetric lidar is similar to traditional aerial lidar, except it uses a green laser that can penetrate water. Under ideal conditions, some systems can collect data at depths up to

50 meters. Bathymetric lidar is limited to clear water with minimal turbulence. Because of this limitation, it is often used along coasts (Figure 4.17) but has limited application along many of the nation's rivers due to high turbidity levels. As the technology continues to advance, application of bathymetric lidar to river systems may become more feasible.
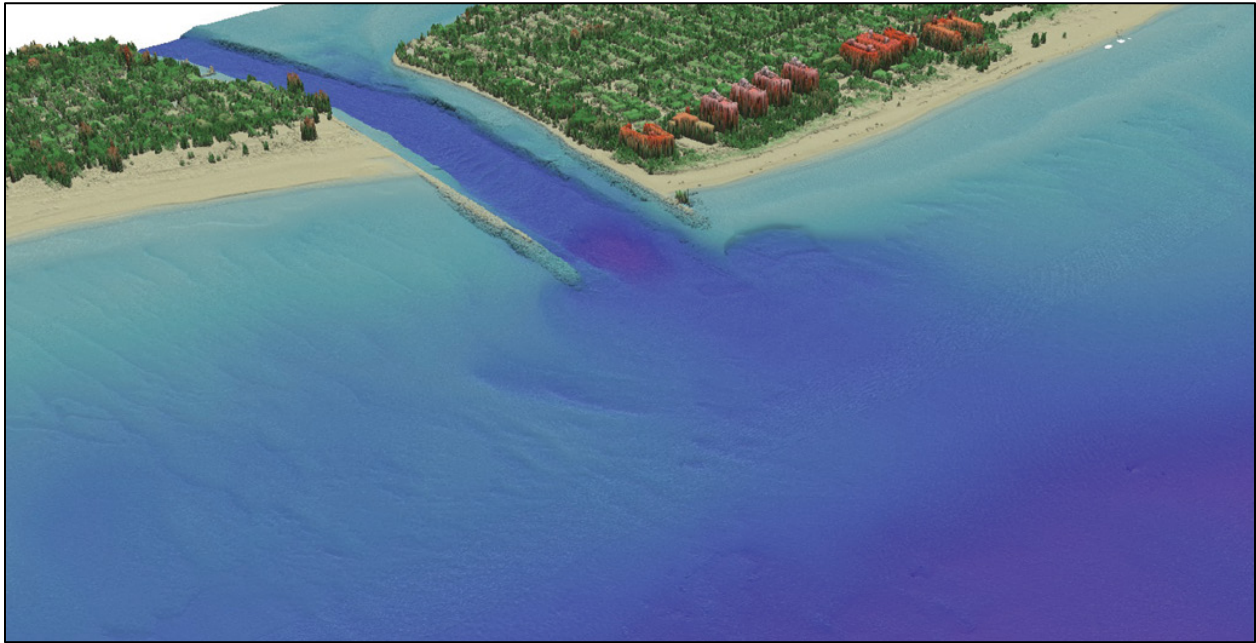
Figure 4.17. Colored contour surface developed from bathymetric lidar data.

Sonar. The most common method of collecting bathymetric data on non-wadable waterways is with sonar, attached to a boat, and combined with a GPS or robotic total station to provide the location of the sonar device. There are multiple types of sonar systems, including single-beam, multi-beam, and swath systems.

Single-beam systems collect a single point, typically every second, as the boat moves across the water. Single-beam surveys typically collect data along cross sections at specified locations or with a specific spacing and may also provide a profile survey along the channel centerline (see Figure 4.18). Rocking of the boat and sensors due to waves, wind, or crew can introduce errors. Correct processing of bathymetric data is important for correct channel representation.

Correct triangulation of single-beam data is essential to create a usable final bathymetric surface (see Figure 4.19). Incorrectly triangulated cross section and profile points combined with topography could result in artificial dams and non-existent high ground within the channel. (see Section 4.2.6 for incorrect triangulation explanation).

Multi-beam and swath bathymetric data are the sonar equivalent of lidar data in terms of coverage (see Figure 4.20). The resulting surface constitutes a nearly complete representation of the underwater surface. Both technologies provide a similar product but, due to different techniques used to obtain data, the swath systems are better suited for shallower waterways.

Figure 4.18. Single-beam bathymetric data points, unprocessed.
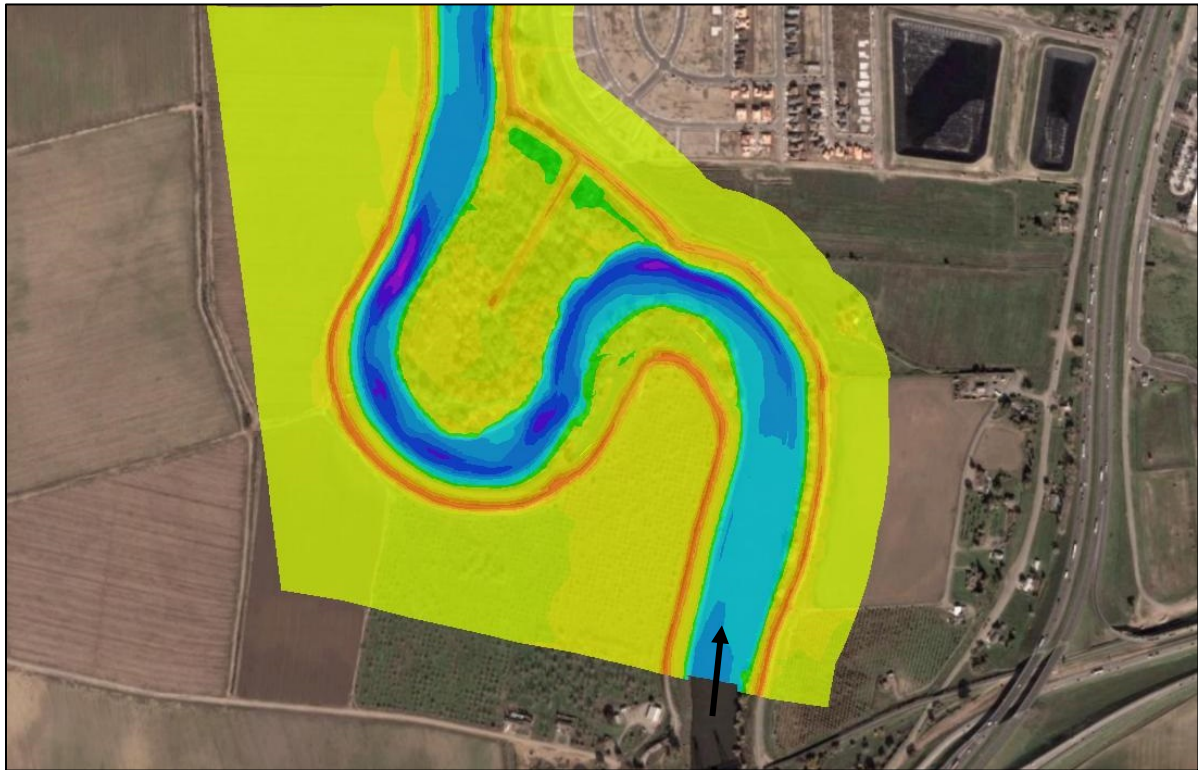
Figure 4.19. Surface triangulated single-beam data combined with topographic data.
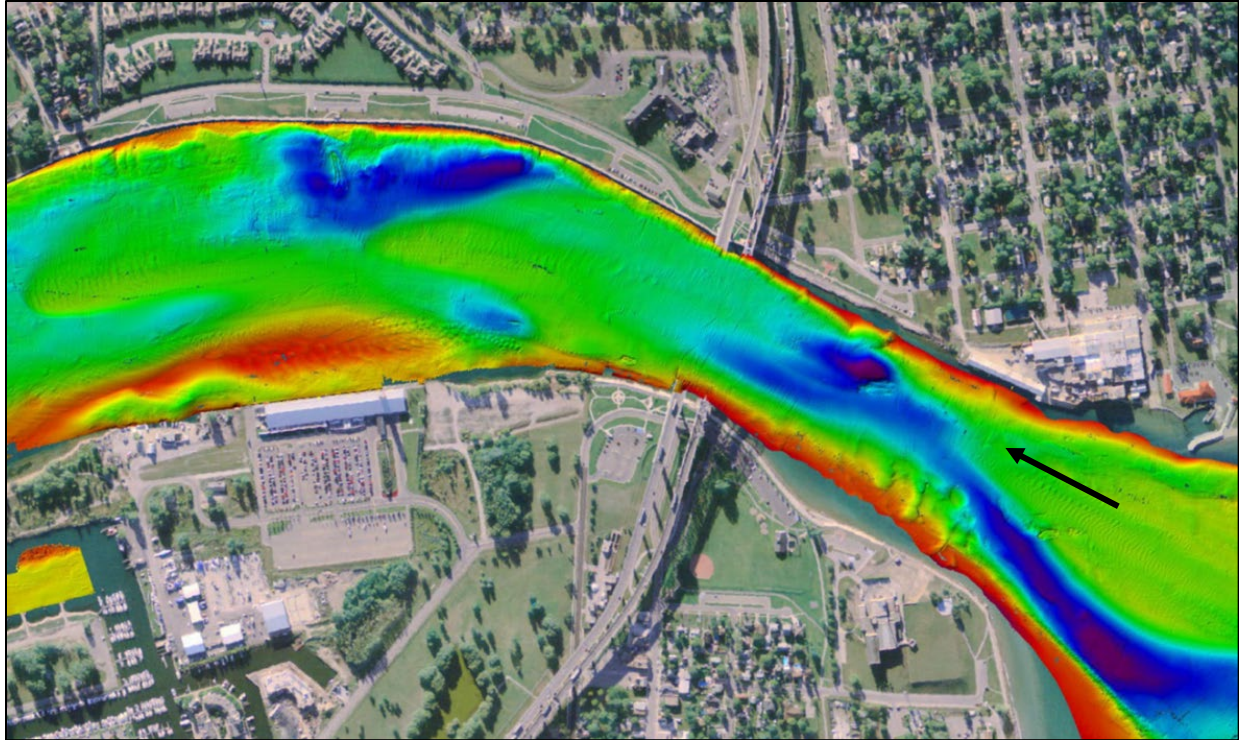
Figure 4.20. Multi-beam bathymetric data surface.

All sonar collection methods have limitations. Sonar does not function in turbulent or aerated water, and sonar systems are not able to collect data in shallow depths (often referred to as a blanking distance). While depths vary, most systems intended for shallow water applications have a blanking distance of 1.0 -1.5 feet. Most sonar systems cannot survey depths shallower than approximately two feet because sonar instruments are typically mounted on a boat below the water surface.

Ground Survey of Streams. As noted above, GPS and total stations can also be used to collect bathymetric survey data of wadable streams. The difference compared to the topographic survey is that the surveyor typically cannot see the variations of the stream bottom. Standard collection methods include taking several cross sections and a profile survey to obtain sufficient data to be able to represent the stream bottom.

Ground-based bathymetric survey data collected in a stream is typically not as dense as data collected above water because of the difficulty of collecting each point. However, it is often feasible to collect enough data to sufficiently map the channel.

As with single-beam sonar, data collected via ground-based methods have to be triangulated correctly to represent the channel. Surveying the edge of water can be very helpful when combining the channel data with topographic data.

Approximated Stream Bathymetry. It is common to have incomplete bathymetric surfaces of channels for an entire 2D model area, similar to 1D model studies. Bathymetric data is typically not readily available for most inland waterways, and schedule or budgetary constraints may limit the amount of data collection for a project. Site conditions may also limit, or prohibit, bathymetric data collection due to safety concerns, turbulent water, or difficult access.

It is standard practice to approximate stream bathymetry when it is not feasible to collect additional channel data. The concept of approximating bathymetric data while also stating that terrain data is the most important component of a 2D model may seem contradictory. However, in the context of having sufficient topographic data with limited or no channel data, it is logical and reasonable to conclude that approximating channel data is an improvement in the representation of the site compared to using the measured WSE, as represented in lidar or photogrammetric data, as the channel. It is best practice to collect detailed bathymetric data in the area of interest and develop approximate data for the reaches that are of less concern.

Several approaches exist that may be used to approximate channel bathymetry. However, three methods are most common. The first option is to develop an average channel section based on channel data from multiple surveyed cross sections from upstream or downstream locations. For instance, if using detailed cross section data near a bridge, an average cross section could be developed and used as a template for other channel locations within the model domain if no survey data exists at those locations. The second option is to develop an approximate trapezoidal channel section representing the channels conveyance capacity developed from the discharge in the channel at the time of the topographic survey and the approximate slope of the channel based on the topographic data. The third option involves creating an approximated channel centerline at a defined offset from the water surface. See Figure 4.21 for examples of the three methods.
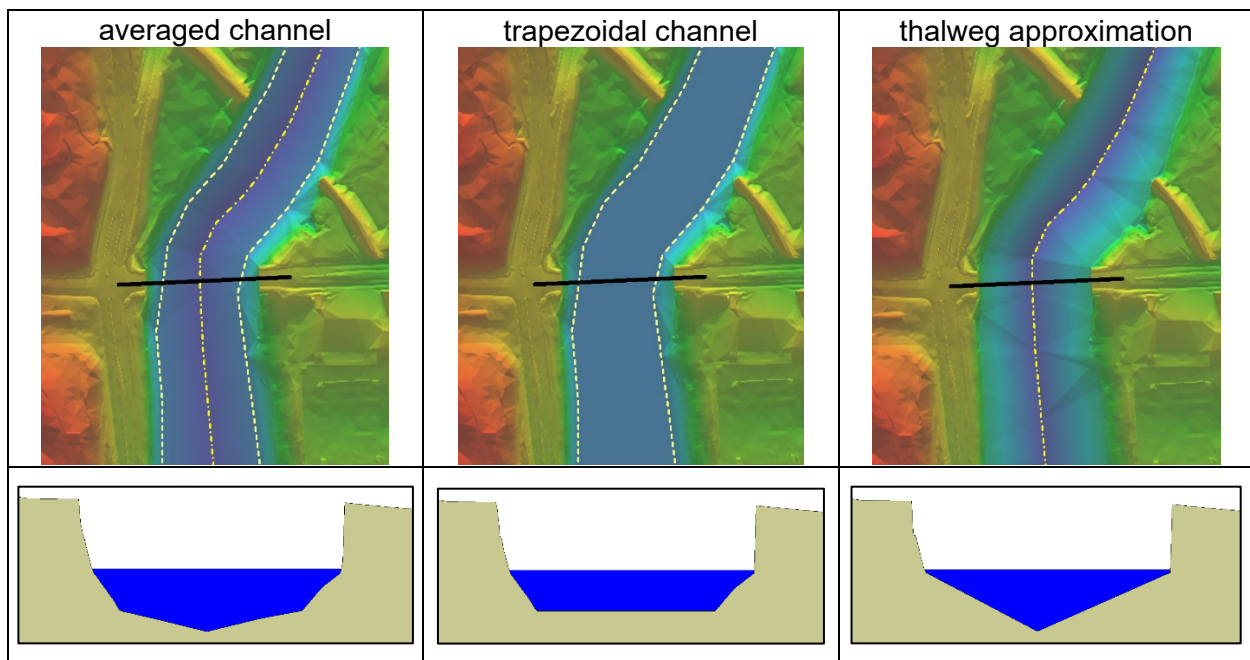
Figure 4.21. Approximated channel bathymetry examples.

All three methods provide an approximate representation of the hydraulic capacity of the channel. It is recommended that more detailed bathymetric collection occur where more detailed hydraulic results are required.

For additional information related to bathymetric mapping, see the following sources:

- USGS – Bathymetric Surveys
- NOAA – Hydrographic Survey Data

Other Terrain Sources: If sufficient data is not available, and it is not feasible to collect additional data from the sources mentioned above, other terrain data options exist. Options provided below are not as accurate as the previously recommended data sources and should be used as a last resort. Using the terrain data sources listed below will result in a 2D model with limitations equivalent to the accuracy of the data.

USGS 10M DEM: Currently, the most widely available topographic data for the US is the 1/3 arc-second data DEM data (approximately 10-meter point spacing). While this data set is available for most of the US, except large portions of Alaska, it has significant limitations for use in 2D modeling. USGS has a goal to collect lidar data for the continental US under the 3DEP, which will become the most up to date data set for the entire US when completed.

Data collection methods and dates can vary significantly across the US, with some data collected in the 1920s. The accuracy of this data set varies due to the date of collection, various collection methods, and sparseness of data points; data can be off by 10-20 feet vertically compared to more accurate elevation data points. Due to the point spacing of this data set, significant hydraulic controls may not be represented adequately (see Figure 4.22).

20-foot contour data (10-meter DEM)

2-foot contour lidar
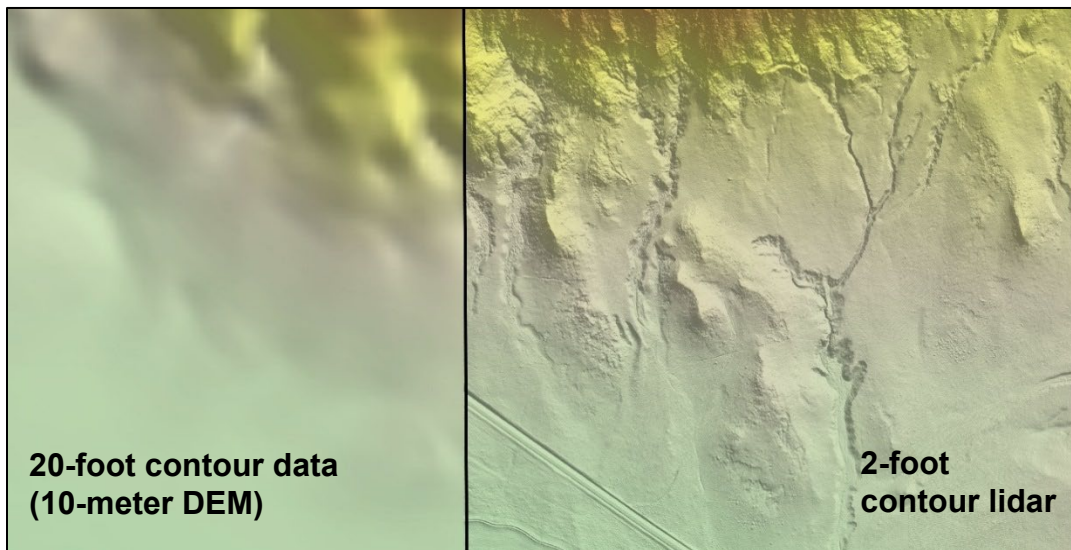
Figure 4.22. Comparison of 10-meter DEM and lidar surface.

IFSAR. Interferometric synthetic aperture radar (IFSAR) mapping technology may be a feasible option for very large and remote locations. Because IFSAR uses radar, it can map topographic surface through cloud cover.

Limitations of IFSAR data are that it does not penetrate dense vegetation as well as lidar, the resulting surface has a lower point density so small topographic features are typically not well represented, and the accuracy is not as good as lidar or photogrammetric collection methods. Currently, the best data available for most of the state of Alaska was obtained using IFSAR, though the USGS is working to improve the accuracy of the topographic data covering Alaska.

Contours. Contour lines represent locations on a surface of equal elevation. Contours are very useful for displaying terrain data on a paper map. USGS topographic maps may be one of the most popular maps produced. They use contours to represent the topography of the entire US.

Contours are derivatives of a terrain surface, which means they are a product of a surface. They are created to help display topographic relief. Contours should not be used to create a terrain surface unless the original terrain surface is not available. This is especially important in areas with relatively flat terrain or small topographic features such as channels or ridges that would not be adequately represented by the contour data.

Because contours are derivative products, or produced from, of a digital terrain model, they are not as accurate as the source data. If contours are used to recreate a surface, the resulting surface would not be consistent with the accuracy standards of the source data and may contain significant errors especially in locations with small channels and relatively flat terrain (see Figure 4.23).
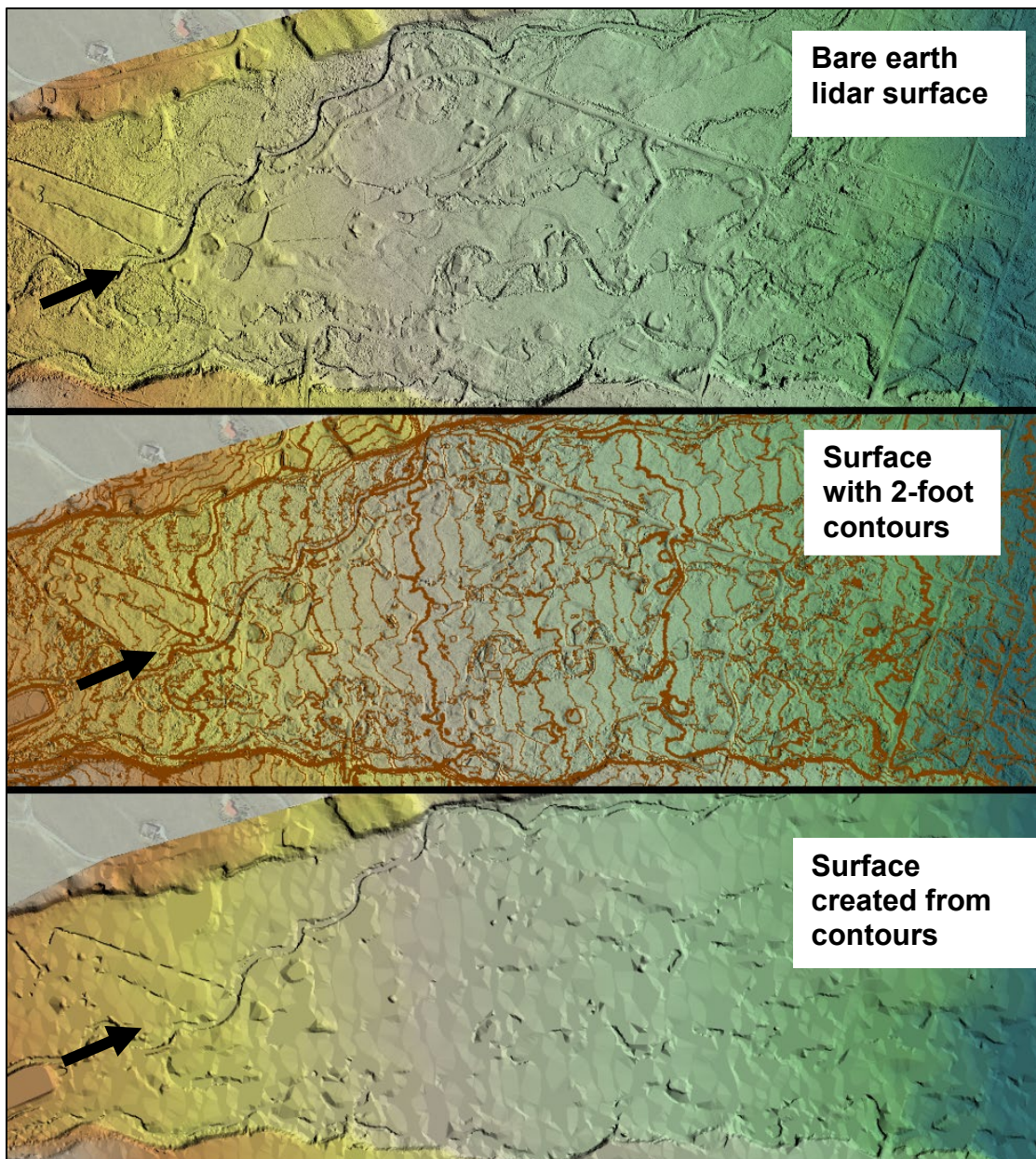
Figure 4.23. Lidar surface, lidar surface with contours, surface developed from contours (note loss of detail).

4.26

4.2.3. Accuracy of Terrain Data

Accuracy requirements for terrain data are typically dictated by the specific model application. Modelers should use the most accurate data available. It is common practice to use terrain data from multiple sources with varying levels of accuracy to create a terrain data set for 2D modeling. End-users of terrain data may not have control over the collection process, but it is important to understand the reported accuracy and the associated limitations of the data.

If additional data is necessary, the requestor should understand the potential range of data accuracies. It is best practice to obtain metadata and available quality control reports that adequately document the reported accuracy of the new or previously collected data. It is the responsibility of the modeler to understand the limitations of the model in relation to the underlying data so that results of the model can be appropriately interpreted.

The horizontal and vertical accuracy of terrain data are both of interest. However, accuracy standards report the vertical accuracy because it has a larger error compared to the horizontal data. If a data set meets a vertical accuracy standard, the horizontal accuracy would be higher than the reported vertical accuracy. The 1947 National Map Accuracy Standards (NMAS) definition that connects the vertical accuracy to the equivalent contour interval provided the commonly accepted language when referring to accuracy as it relates to terrain data (i.e., 1-foot contour and 2-ft contour data). Equivalent contour interval correlates a standard contour interval and terrain surface accuracy of one-half the contour interval. So, a 1-foot contour equivalent data set has a vertical accuracy of ±0.5 feet.

The 1998 National Standard for Spatial Data Accuracy (NSSDA) replaced NMAS for all digital mapping products. While the NMAS stated accuracy does not apply to the standard collection methods used today, it is helpful to provide values for comparison purposes (see Table 4.4).

The Positional Accuracy Standards for Digital Geospatial Data (ASPRS 2014), as published by ASPRS, is the most comprehensive and widely adopted standard for the geospatial industry. The society has developed topographic mapping standards that focus on absolute surface accuracies that are more relevant to contemporary technologies.

The following tables provide reported accuracies for the most common terrain data types used to develop 2D models. References for accuracy values are provided below each table. Standard data collection methods include ground survey & digital mapping products (Table 4.4), lidar (Table 4.5), and photogrammetry (Table 4.6). Bathymetric data accuracy can vary significantly because of various collection methods, cross section spacing, point spacing, and proper triangulation of points and the use of breaklines. For more information related to hydrographic survey standards, see the USACE requirements (USACE 2017) or the International Hydrographic Organization standards (IHO 2008).

Several terms and definitions are necessary to communicate the range of accuracies reported for different collection methods. Some terms are specific to only one type of collection method. This document includes definitions and discussions of some terms, though it is not in the scope of this document to thoroughly explain all the terms. Refer to the references provided for complete definitions and a more in-depth explanation of accuracies and related terms.

Table 4.4. NMAS and NSSDA standards.

NMAS Equivalent Contour Interval (ft)	NMAS Vertical Accuracy Standard 90% Confidence Level (ft)	NSSDA RMSE (ft)	NSSDA Vertical Accuracy at 95% Confidence Level (ft)
1	0.5	0.3	0.6
2	1.0	0.6	1.2
5	2.5	1.5	3.0
10	5	3.0	6.0
20	10	6.1	11.9
RMSE, root mean square error in the z-direction			
Source: https://www.fema.gov/media-library-data/20130726-1752-25045-1006/elevations_appb.pdf			

Table 4.5. Absolute vertical accuracy for lidar digital elevation models.

Quality Level	RMSE, (non-vegetated) (ft)	NVA at the 95-percent confidence level (ft)	VVA at the 95th percentile (ft)	Aggregate nominal pulse density (pls/m^2)
QL0	≤0.16	≤0.32	≤0.49	≥8.0
QL1	≤0.33	≤0.64	≤0.98	≥8.0
QL2	≤0.33	≤0.64	≤0.98	≥2.0
QL3	≤0.66	≤1.29	≤1.97	≥0.5
QL, quality level; NVA, non-vegetated vertical accuracy; VVA, vegetated vertical accuracy				
Source: https://pubs.er.usgs.gov/publication/tm11B4				

Table 4.6. The absolute vertical accuracy of the new ASPRS 2014 standard compared with legacy NMAS standard (ASPRS may apply to photogrammetric or lidar data).

Vertical Accuracy Class (in)	RMSE Non-Vegetated (in)	Equivalent contour interval per NMAS (in)
0.39	0.39	1.30
0.98	0.98	3.24
1.97	1.97	6.48
3.94	3.94	12.95
5.91	5.91	19.43
7.87	7.87	25.91
13.11	13.11	43.13
26.26	26.26	86.39
39.37	39.37	129.91
131.22	131.22	431.69
Source: https://www.asprs.org/a/society/committees/standards/Positional_Accuracy_Standards.pdf		

Because lidar data is the most common type of topographic data available in the US, modelers should become familiar with the details of the quality levels (QL). While QL standards are not identical to the commonly used NMAS contour interval standards, comparisons are helpful in understanding reported accuracies. Quality level 1 and 2 are comparable to the NMAS 1- and 2-foot equivalent contour interval standards, though as seen in the tables above, QL 1 and 2 have higher vertical accuracy. The tables only report part of the standards for each QL. While the QL 1 and 2 data have the same accuracies, QL 1 data has a higher point density and therefore provides more detail for smaller topographic features which is beneficial for areas with low relief since smaller features can have a more significant impact on hydraulics.

Specific geospatial data accuracy standards may apply to some projects depending on the geographic location, regulatory agencies involved, and model application. For example, FEMA related hydraulic modeling projects need to meet FEMA's vertical accuracy requirements. Most FEMA related projects require QL 2 or 1 data. For transportation projects where floodplain impacts are the focus, FEMA mapping standards are acceptable. Modelers should determine if specific accuracies are required for their models.

In addition to the reported accuracy of the data, modelers should also consider the date of the survey. River systems are dynamic, and the terrain within the channel and floodplains change over time due to natural or human actions. The date of the data is of higher importance if the terrain of the project location has changed significantly over time. As noted previously, it is the modeler's responsibility to determine if the terrain data represents the project site sufficiently given the intended application of the model.

4.2.4. Typical Terrain Data Formats

Digital representations of terrain include a combination of points, lines, and areas. The points are triangulated together to create a surface, with breaklines forcing specific triangle edges. Enclosed areas can be included to limit triangulation of data within, or outside of, designated locations. All data from different survey methods, surfaces within various software applications, and file formats can be broken down into these basic parts.

Terrain data can be provided in individual files representing the points, lines, and areas, or as triangulated surfaces. Surfaces can include grids or triangulated networks. Terrain data can also be delivered and viewed within common software applications such as CAD or GIS. Common terrain data files and surface formats are provided in Table 4.7.

Table 4.7. Common terrain data formats.

Format	Definition	Comments
tin	Triangulated irregular network	Triangular faces connecting survey points and enforced by breaklines. TIN files are typically software specific and are often not interchangeable between various software packages
raster (grid)	A grid of squares used to represent various data types	Data is organized by an origin coordinate value, equal grid size, and a value for each grid (elevation or color for example)
dem	Digital elevation model	Term used to refer to digital mapping data; also, a specific raster file format
dtm	Digital terrain model	Term used to represent the bare ground; also, the digital surface file format used by Bentley InRoads
dsm	Digital surface model	A term used to represent all natural and manmade features, similar to lidar first return data or unprocessed UAS data
USGS dem	Raster based dem developed by USGS	A raster-based digital elevation model
flt	Float grid file	Common grid file type used for digital elevation data
img	ERDAS imagine file	Common grid file type used for digital elevation data
grd	ESRI grid file	Common grid file type used for digital elevation data
tiff	Tagged image file format	Raster data image file format, can also store elevation data
las	Binary file 3D point cloud	Primarily used for lidar data, includes more than the x,y,z data of each point; created as an improvement to the generic ASCII file type
laz	Compressed las file	Provides a way to reduce las files; may need to be unzipped before using in some software applications
xyz	3D points	Basic text file typically in comma or space delimited format that can be opened in a spreadsheet application and most geospatial software programs
ascii	Standard code for point information	Similar to xyz data but can contain other information like structure geometry, point ID, point type, etc.

4.2.5. Processing of Terrain Data

Two-dimensional modeling projects typically involve obtaining, reviewing, and merging various terrain data sets. The workflow diagram presented in Figure 4.24 provides a standard approach to creating the necessary data to create a 2D model. The steps illustrated in the workflow diagram are discussed below.
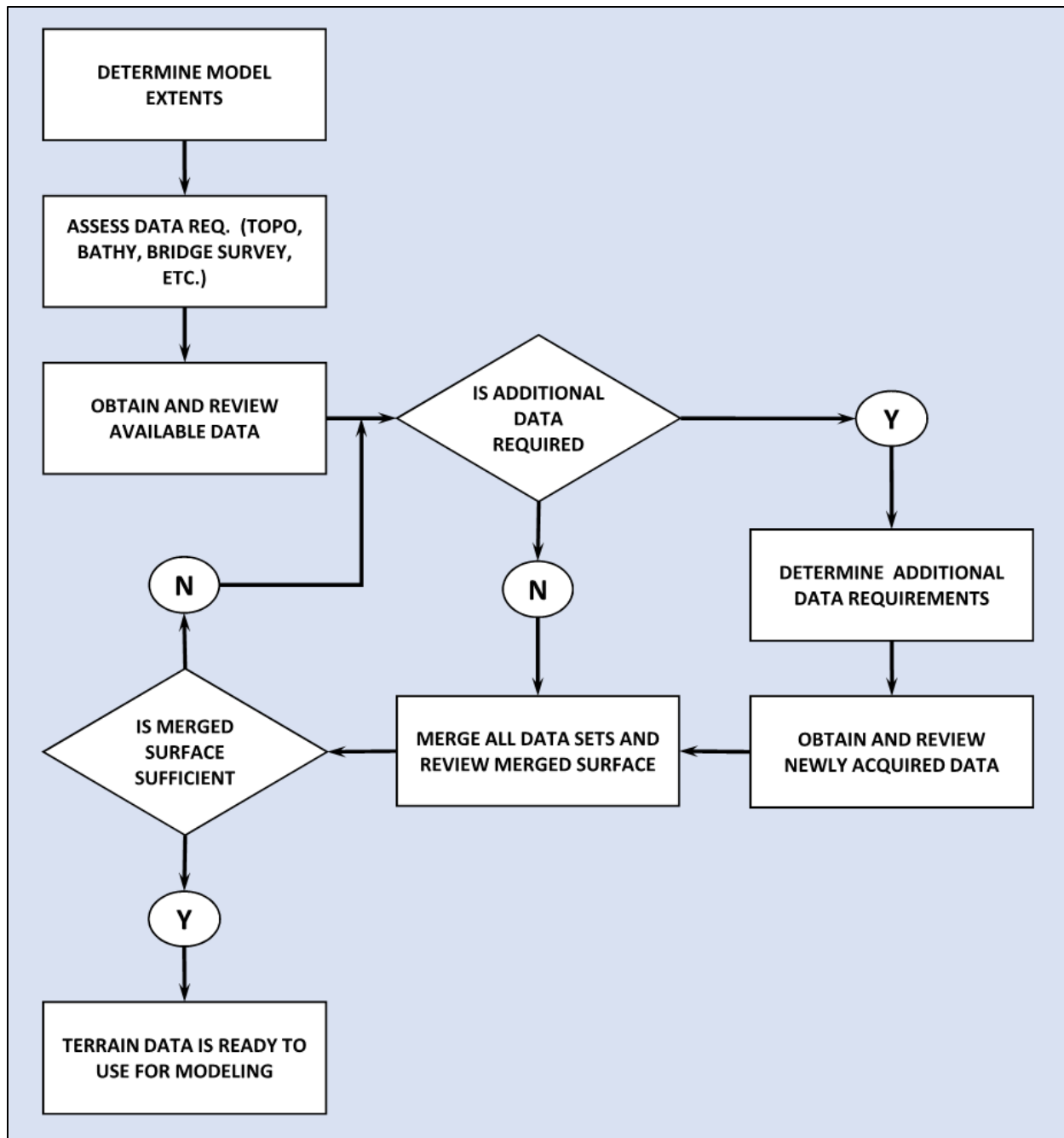
Figure 4.24. Terrain data creation workflow.

Determine model extents – Terrain data should be collected for the area to be represented by the 2D domain. See Section 5.2 for more details.

1. Assess data requirements – Determine terrain data required for a model such as topographic, bathymetric, and roadway/bridge as it relates to the terrain.

2. Obtain and review available data – Research online sources from federal, state, county, city, and local consortiums (associations of government agencies). Determine answers to question

such as; Is the data adequate, does it sufficiently represent the project site, are hydraulic controls represented, and is the data recent enough? See Section 4.1.1 for potential sources.

3. Determine additional data requirements – Determine if additional terrain data is required to represent the project site.

4. Obtain and review additional terrain data – See Appendix B for site survey request form. Review data as described in Step 3.

5. Merge all data sets and review surface – Combine all data sets into one surface. Verify there are no discrepancies between different data sets, resolve any differences as necessary to represent the project site.

4.2.6. Terrain Data Issues

When reviewing terrain data, locations to check due to common issues include:

- Areas under bridge decks
- Dense vegetation
- Bathymetric data within the river channel

Aerial lidar and photogrammetry cannot "see" under bridge decks. Processed bare earth lidar data should have lidar points representing bridge decks classified as such and removed. The triangulation of the bounding points typically misrepresents the topography under the bridge deck (see Figure 4.25). Modelers should verify that bridge decks have been removed and determine how much additional survey data should be collected to represent the ground under the bridge deck adequately. Bridge openings, and associated roadway, typically act as hydraulic controls and should be correctly represented in a 2D model.

All survey methods have difficulty fully representing areas with dense vegetation. It may not be feasible for a modeler to determine if terrain data truly represents the ground. However, it is a good practice to review terrain data in locations with dense vegetation to determine if the final surface is reasonable. The terrain data should be reviewed to verify the removal of lidar points representing vegetation and that hydraulic controls are appropriately represented (see Figure 4.26 and Figure 4.27).

As noted earlier, most topographic data collection methods do not obtain data underwater. At a minimum, the modeler should review the channel area to verify: (1) that hydro-flattened breaklines have been included, (2) if the channel was dry, or low enough, at the time of collection to not need bathymetric data, and (3) that the lidar data reasonably represents the channel geometry. If additional channel survey data is required, then the bathymetric survey data has to be merged with the topographic data. Ideally, topographic and bathymetric data sets should overlap to assist in merging the data. Incorrect triangulation of bathymetric data points and at the interface of these two data sets is common (see Figure 4.28). The modeler should review the triangulation to verify that it is reasonable and does not create erroneous high points in the channel.
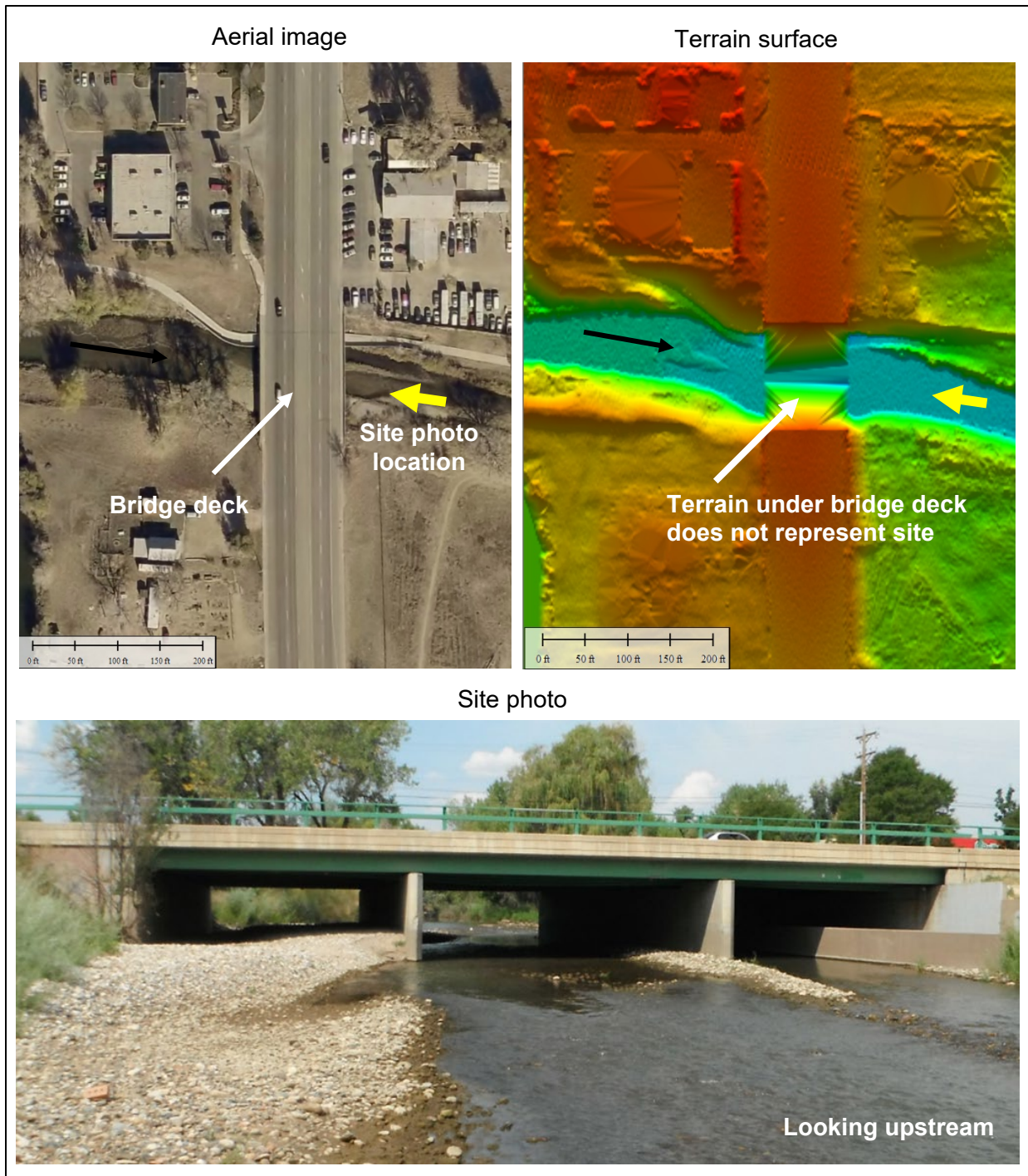
Figure 4.25. Aerial image of site, bare earth with bridge deck removed showing incorrect topo and photo of bridge.
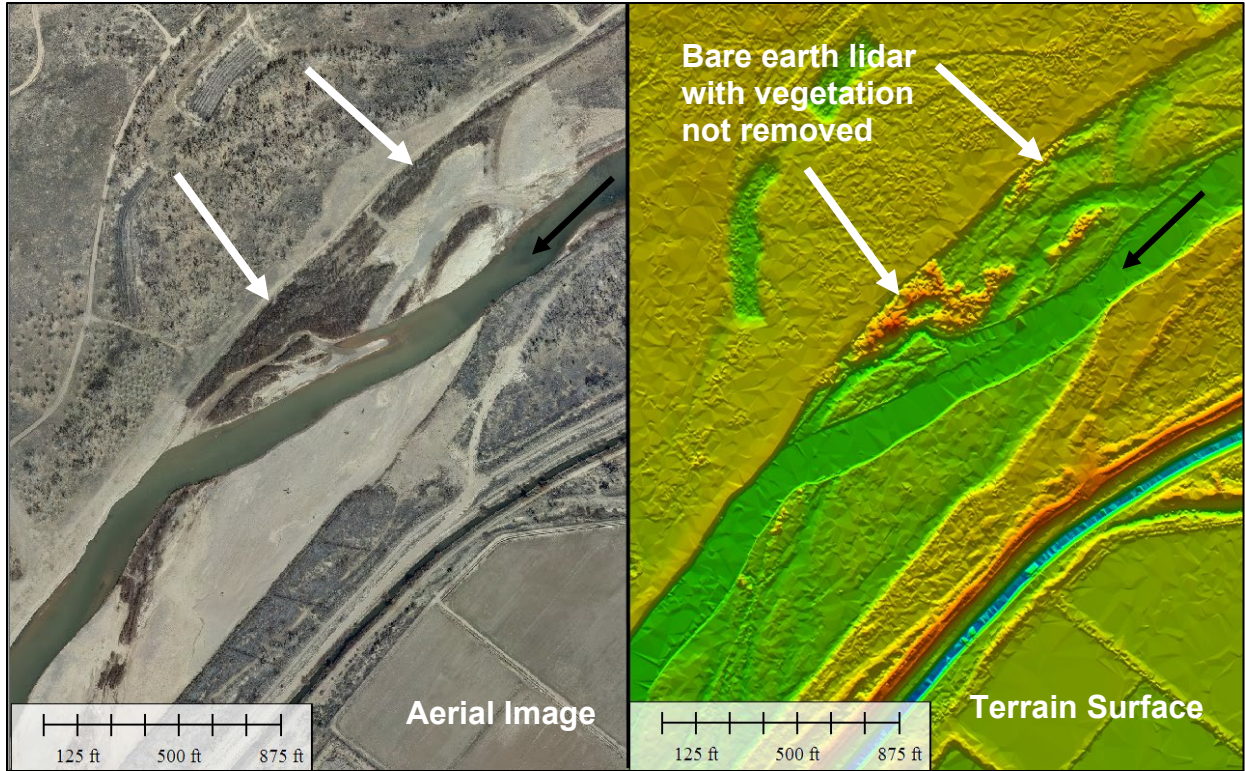
Figure 4.26. Bare earth lidar that still includes data representing vegetation.
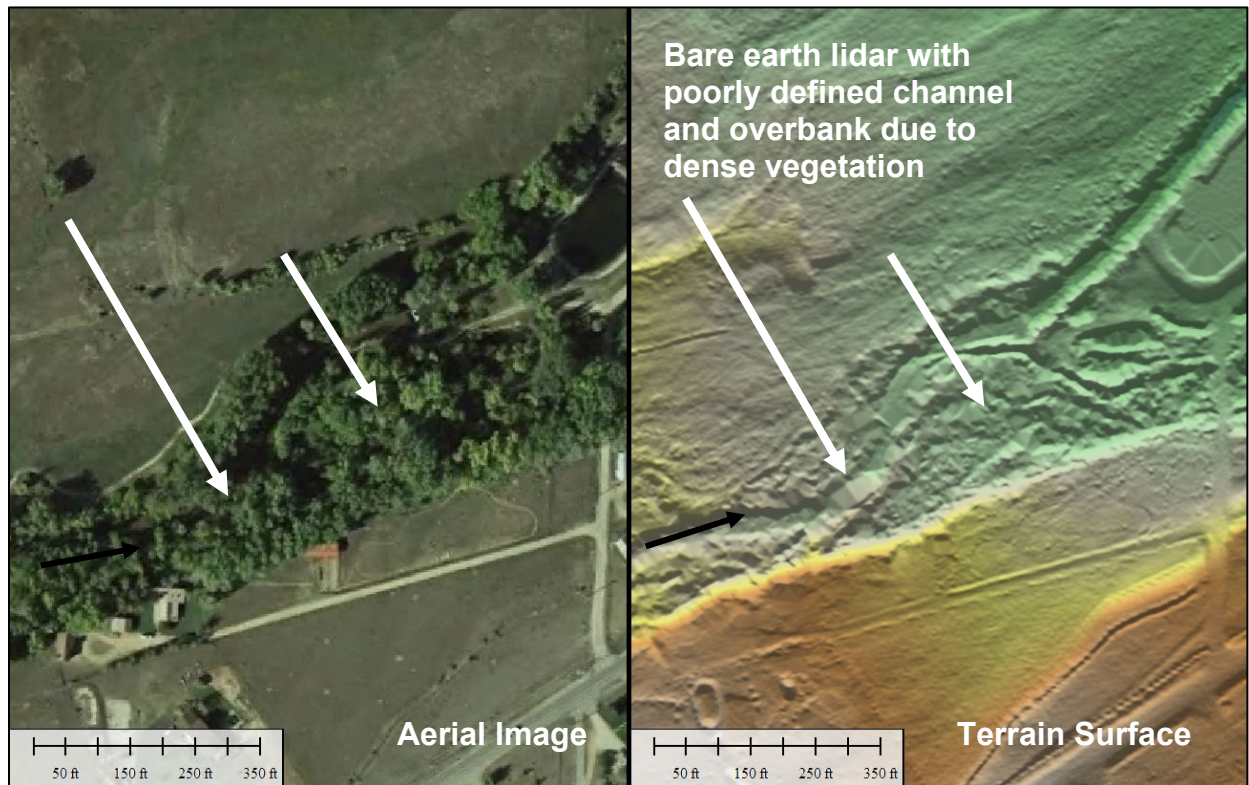
Figure 4.27. Bare earth lidar with poorly defined channel and floodplain due to dense vegetation.

4.34

Overbank breaklines and bathymetric points	Incorrectly triangulated points	Contours with artificially high areas protruding into channel

Added bathymetric breaklines	Correctly triangulated points	Contours representing channel bathymetry
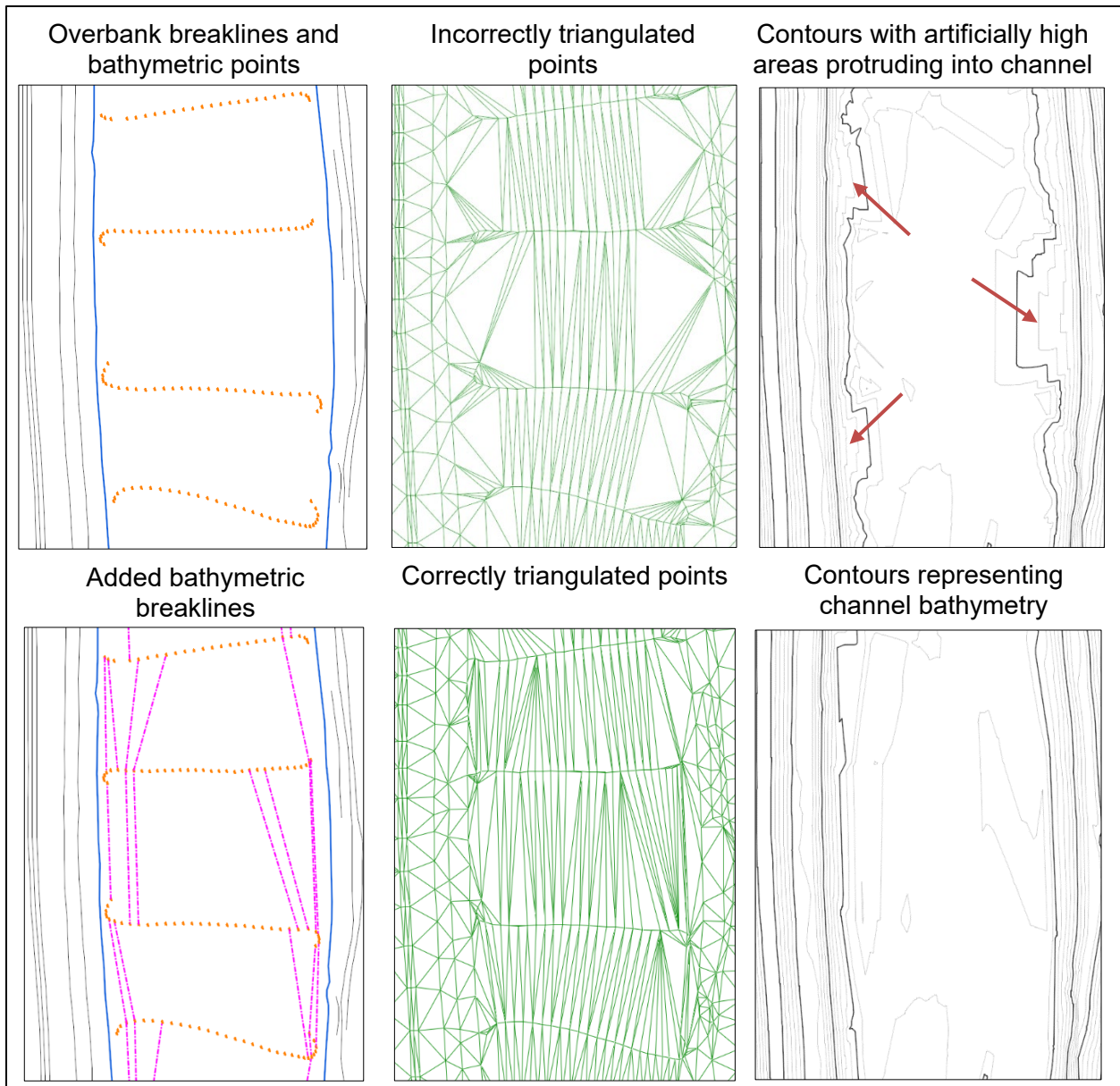
Figure 4.28. Bathymetric points incorrectly triangulated and properly triangulated.

4.3.   BOUNDARY CONDITIONS

Data for boundary conditions are available from a variety of sources. Different data sources may provide a range of accuracies. It is best practice to review boundary condition data and sources for reasonableness and applicability for each project location.

Boundary conditions can be categorized either as external or internal. External boundary conditions are defined along the edges of the model extents where water flows into and out of a model domain. For steady flow conditions, the most commonly required input for external boundary conditions is peak discharge at the upstream boundary and a corresponding WSE at the downstream boundary. For unsteady flow conditions, commonly used inputs include discharge hydrographs at the upstream boundary and rating curves at the downstream boundary.

Internal boundary conditions are specified inside the model domain, away from the edge of the model boundary. Examples of internal boundary conditions include hydraulic controls such as stormwater drain and outlets, weirs, and culverts, as well as internal inflow or outflow locations, sometimes referred to as sources or sinks, respectively.

Some models have the ability to include rain, or other physical processes, as a boundary condition. This capability is often referred to as "rain-on-grid" (see Section 5.4.5). These types of boundaries are applied over a portion, or the entire area, of the model domain. Typical applications of these types of boundary conditions include complex urban applications or watershed hydrology studies.

4.3.1. Upstream Inflow

Common methods for estimating discharges used in engineering projects include statistical analyses, regional regression equations, USGS stream gauge analysis, and more detailed hydrologic modeling. Depending on conditions in the basin, availability of information, and specific requirements for the project, any one of these methods may provide a reliable estimate of discharge.

HDS-7 lists the following sources of peak discharge, in general order of preference:

- Prior flood frequency studies by authoritative sources
- Statistical frequency analysis of gauge records at the site
- Gauge analysis transferred from a nearby gauged location to an ungauged location
- Applicable regional regression equations
- Hydrologic models

Prior flood frequency studies from FEMA, USGS, USACE, or analyses conducted by state or local agencies are good resources. If using prior studies, the date of the study should be assessed to determine if the study is still applicable. Gauge record analysis at the gauge site, or transferring a gauge analysis from a similar nearby basin, can also be a source of information if gauge record lengths are 10 years or longer (according to USGS). However, 10 years may not necessarily be long enough to give a reliable estimation, particularly for larger flow events. Regional regression equations are an option when no suitable gauge sites exist for the basin or similar nearby basins. The accuracy of regional regression equations can vary greatly depending on the region and the degree of similarity between the basin of interest and similar basins analyzed for the development of the equations. For basins where regional regression equations are not trusted and for highly regulated basins, hydrologic modeling may be required (FHWA 2012b).

The USGS StreamStats web-based application computes streamflow statistics for gauged and ungauged sites throughout the US. This site is a helpful source for peak flow estimation, or as a ballpark check for other methods of determining model hydrology. The HDS-2 manual (FHWA 2002) provides additional information on the aspects of hydrological analysis for highway engineers.

4.3.2. Downstream WSE

A common downstream boundary is a known WSE. A single WSE does not typically represent reality in natural river systems, though it is a valid approximation. The WSE can be a single value or represented by a rating curve relating WSE to discharge. Common resources of WSE's include FEMA Flood Insurance Studies (FIS), gauge data from the USGS, and observed high-water

marks when the corresponding discharge is known or can be estimated. Previous 1D and 2D models can be sources of WSEs if available.

Normal depth can also be used as an approximation of WSE if known values are not available. In natural channels, flow conditions are unlikely to satisfy normal depth criteria due to longitudinal topographic and roughness variations. However, normal depth is a reasonable approximation if more representative data is not available, but the modeler should understand the potential hydraulic impacts of this assumption at the project location. Many hydraulic modeling software applications have the ability to compute the normal depth using an energy slope and discharge courtesy of the modeler. Energy slope can be calculated from the average channel slope determined from the terrain data, or the floodplain slope of the WSE if available.

Critical depth is an option if the exit is guaranteed to be supercritical such as when a control structure produces a sudden drop in the channel or steep channels, but critical depth is not common in most natural alluvial channels.

Rating curves can be developed from the sources mentioned above for a range of discharges. Some software applications provide additional capabilities in developing rating curves from normal depth and critical depth calculations, including 1D models. There are situations when a FEMA FIS only includes the 100-year profile, or only includes flood discharges that do not match the desired discharge. When confronted with this situation, the energy slope can be computed from one discharge and applied to other discharges. This approach should be used in conjunction with extending the model further downstream (FHWA 2012b), or a sensitivity analysis (see Section 6.5).

4.4. HYDRAULIC STRUCTURES

Hydraulic structures such as bridge decks, piers, culverts, weirs, and gates are often required for 2D models, either as the subject of analysis or as hydraulically significant features. Each type of structure has specific data requirements necessary for modeling. Not all models handle hydraulic structures to the same extent. Modelers should consider the level of analysis required for the structures when choosing which model to use (see Section 2.5).

Structures can be represented in the model geometry, or as internal controls within the mesh. When using internal control methods, multiple lines, or an area, controlled by user inputs, typically represent structures. Input data for hydraulic structures can be obtained from a variety of sources, including survey data (see Appendix B), record drawings, design plans, aerial imagery, site visits, and inspection reports. General data requirements for structures may include:

- Existing and proposed structure geometry (length, width, diameter)
- High and low chords of bridges
- Inlet and outlet types for culverts
- Presence of vegetative debris
- Structure material

Bridge Deck. A bridge deck should be included in a 2D model if the water surface is expected to be at or above the bridge low chord elevation, creating a blockage to flow and potentially resulting in pressure flow. When pressure flow occurs, the WSE becomes constrained by the deck causing the velocity to increase through the bridge opening. Data input required for a bridge includes:

- Low chord geometry (length, width, and shape - flat, angled, arched, other)
- Deck thickness (to determine high chord/overtopping geometry if applicable)

Some models have the ability to account for discharge overtopping the bridge deck through a 1D weir flow calculation. If required, the length of the overtopping area, crest elevation, and weir type coefficient are also needed. Bridge railings may need to be modeled as high chord elevation, especially if debris is likely to accumulate. If a bridge is expected to experience pressure flow and overtopping flow, the hydraulics associated with this type of situation would be highly dynamic and 3D. Two-dimensional approaches are only capable of approximating the hydraulics for this type of complex situation.

Piers. Piers can be represented in 2D models by multiple methods, for more information, see Section 5.5.1. Piers can be simple single circular columns or can include a combination of multiple piles, a pier cap, and a pile bent. Data requirements for piers include:

- Pier geometry (diameter, length, thickness, type)
- Pier component elevations for complex piers

Culverts. Specific data requirements may vary depending on the chosen software and model, and how the modeler has determined to represent the culvert within the model. For instance, a detailed analysis of a large culvert requires more data than a small culvert that has only a minimal impact on the project. Culvert data requirements may include:

- Culvert geometry (culvert type, size/dimensions, length, slope, end treatment)
- Associated roadway geometry for overtopping calculations

HY-8, developed by FHWA, provides common tools for analyzing culvert hydraulics and can be used in conjunction with a 2D model.

Weirs. A weir is a barrier across a flow path that alters flow characteristics. Common types of weirs include broad-crested or sharp-crested. Most hydraulic models can effectively simulate broad-crested weirs, such as roadways, using the mesh geometry. For sharp-crested weirs, it may be challenging to generate a mesh with sufficient resolution to effectively represent flow conditions at the weir. Therefore, the weir equation would provide a better representation of the hydraulics across the weir.

A common approach for representing a weir is with two arcs, one for the upstream face and the other for the downstream face of the weir. The weir equation determines the hydraulic variables at each arc. The computed discharge is removed from the model at the upstream arc and introduced again at the downstream arc. Typical model inputs required for the weir equation include:

- Crest elevation
- Length of the weir
- Weir coefficient based on weir type (see HDS-7 for further information)

Gates. Typical examples of gates include sluice gates and Tainter (radial) gates. A gate boundary condition can account for flow both over and under the structure. Data requirements for gates include:

- Gate geometry (crest elevation, gate opening height, and gate opening width)
- Contraction coefficient for calculations under the gate
- Weir coefficient for water flowing over the gate

4.5.   LAND USE TYPES AND ROUGHNESS ASSIGNMENTS

This section provides information to assist modelers in determining the friction loss coefficients for 2D models. The focus is on roughness parameters for flow in natural channels and floodplains. The Manning roughness coefficient (Manning, 1889) is the most common method used in the US of accounting for resistance to flow from the material water is flowing over. The Manning coefficient "n" is commonly used to represent land cover and roughness for both 1D and 2D hydraulic models.

4.5.1. Roughness Estimation

For 2D models, roughness values are assigned to every mesh element based on various land use activities, typically as delineated from an aerial image (see Figure 4.29). Depending on the model and software used, roughness values may be set by spatial polygons (GIS data) representing various land use types or assigned manually per mesh element. Either way, each element is assigned a specific roughness value.

Determining roughness values from aerial imagery, in combination with a site visit and site photographs, is the most common method for determining various land use types. Land use information available online can be helpful. However, it may not contain adequate resolution for some modeling applications, including hydraulic analysis related to transportation infrastructure.

Areas of similar land use types delineated at an appropriate scale for the application of the model need appropriate Manning's n values to represent hydraulic roughness. Several resources, provided in the list below, are available to assist modelers with estimating reasonable Manning's n values.

- Guide for Selecting Manning's Roughness Coefficients for Natural Channels and Flood Plains (USGS 1989) – Water Supply Paper 2339

- HEC-RAS River Analysis System Hydraulic Reference Manual (USACE 2016) – Readily available 1D hydraulic reference manual with Manning's n value table.

- Open Channel Hydraulics (Chow 1959) – Extensive resource on open channel hydraulics, including a widely used table for Manning's n values.

- Roughness Characteristics of Natural Channels (USGS 1967) – Water Supply Paper 1849

- Determination of the Manning Coefficient From Measured Bed Roughness in Natural Channels (USGS 1970) – Water Supply Paper 1898-B

- Estimation of Roughness Coefficients for Natural Stream Channels with Vegetation (USGS 1998) – Water Supply Paper 2441

- Flow Resistance Estimation in High-Gradient Streams (Yochum, 2010)

- [Determination of Roughness Coefficients for Streams in Colorado](#) (Jarrett 1985)

- Estimating Hydraulic Roughness Coefficients: Agricultural Engineering (Cowan 1956)
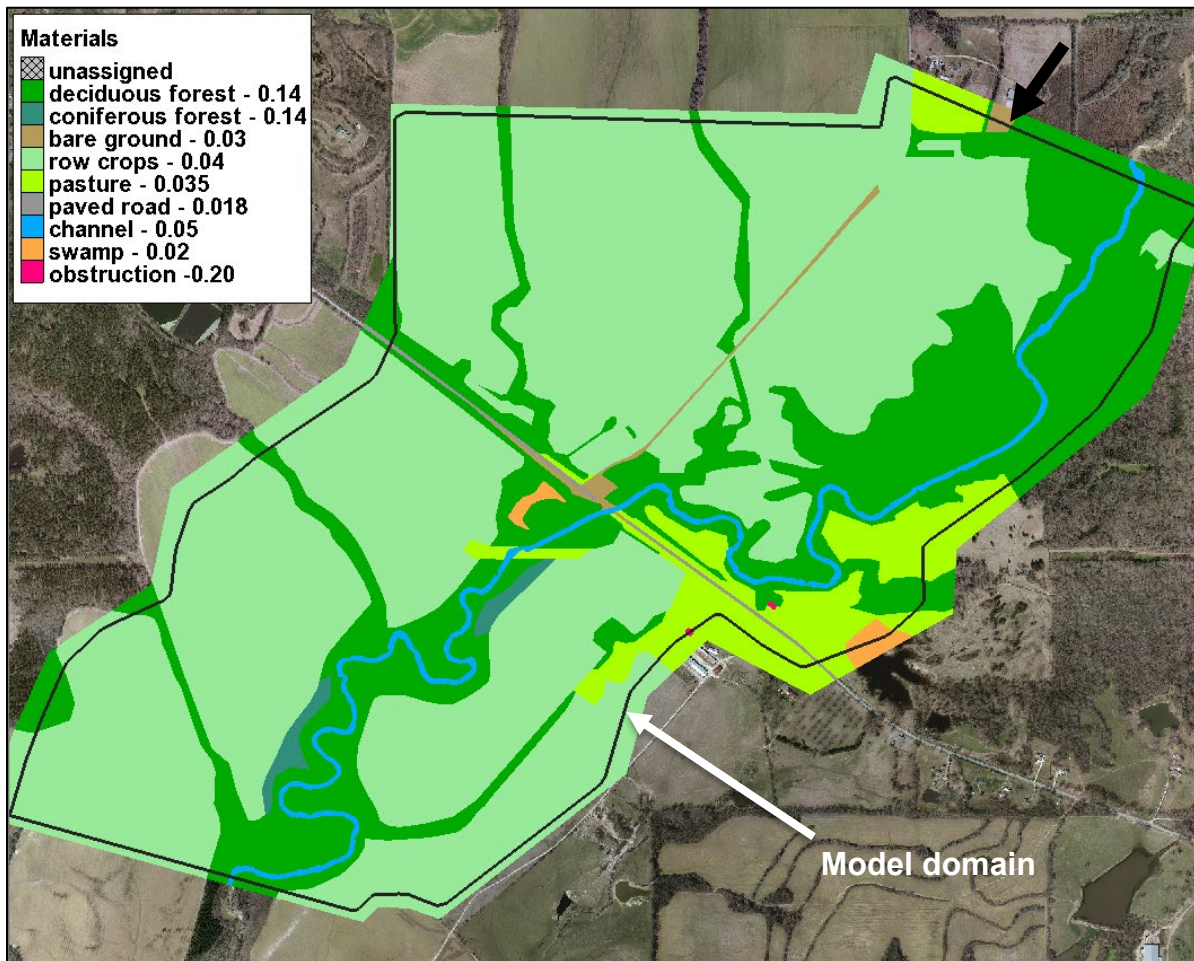
Figure 4.29. Example land use type designation and associated roughness values.

4.5.2. 1D vs. 2D Roughness Parameters

Roughness values provided in the resources cited above were developed in the context of using simplified equations and 1D models. For example, Cowan (Cowan, 1956) published the following relationship for estimating Manning's n value, referenced in WSP 2339:

$$n = (n_o + n_1 + n_2 + n_3 + n_4)m_5 \qquad (4.1)$$

This equation uses terms that account for the degree of irregularity (n_1), the frequency of variation of channel cross section (n_2), and the degree of meandering of the channel (m_5). In a 1D model, these terms play an important role in developing the roughness coefficient, primarily because a 1D model is dependent on cross section placement assumptions and frequency of placement and assumes no lateral movement of flow.

In a 2D model, the shallow water equations or the representation of the terrain by the mesh, directly account for these terms. Because of this, it is reasonable to conclude that the roughness

values used for 2D models should be lower than 1D models. Calibrated 2D models have demonstrated that Manning's n values for 2D models are approximately 10% less than values used for calibrated 1D models. However, there have not yet been sufficient studies published on this topic to make definitive conclusions.

FHWA recommends using unadjusted Manning's n values from the resources provided above for estimating roughness values for use in 2D models. Determining appropriate Manning's n values is an estimation usually based on modeler judgment and experience. The current standard of practice does not result in Manning's n values accurate enough to justify making minor adjustments to recommended values. For example, Chow's (Chow, 1959) recommended range of Manning's n values for a clean, winding channel with some pools and shoals has a minimum value of 0.033, a typical value of 0.040, and a maximum value of 0.045. Applying a 10% adjustment to the typical value results in an adjusted value of 0.036. Because this small adjustment is beyond the level of accuracy achievable in estimating Manning's n values, it is sensible to continue to use the values provided in standard resources. When sufficient calibration data is available, reasonable adjustments to estimated values may be justified. If modelers use Cowan's method provided in equation 4.1 above, it is not necessary to include the n_1, n_2, and m_5 terms.

4.5.3. Depth Varied Roughness Values

Most 2D models allow for depth varied Manning's n values, similar to 1D models. Many land use types can result in a variation of roughness values depending on flow depth. Chow presented results from various studies that define how the Manning's n value decreases with an increase in depth and discharge (Chow 1959, pp 104-106). However, there is only limited information presented by Chow for the numerical values of n related to different depths. During laboratory experiments, the relative roughness of willows standing and laying down during different discharges determined (NCHRP 2016, pp 70-72), but the range of flow depths and willow configurations are limited (see Figure 4.30).

Depending on the 2D model used, a minimum of two Manning's n values are required when applying a depth varied Manning's n. Some models may allow for more than two n value points. The modeler should understand how the specific software and model interpolate data between defined n value depth points. The use of depth varied Manning's n values is reasonable and may be beneficial to some 2D model applications; however, there is currently very little reference information available. Therefore, it is the responsibility of the modeler to determine appropriate depth-varied values based on judgment and experience.

Figure 4.30. Calibrated n values for willows standing and lying down in flowing water.

4.6. ADDITIONAL BACKGROUND AND REFERENCE DATA

Additional background data may be helpful during the development, calibration, and plotting of results from a 2D model. Common background data includes aerial imagery and various data sets in CAD or GIS files such as land use, hydraulic structure linework, future conditions information, and WSE.

Geospatial data that may be helpful with 2D modeling exists with a broad spectrum of accuracies (See section 4.5.2) and is accessible via GIS and CAD software. While data with varying levels of accuracy can be in either type of software, it is common that most GIS data is not as accurate as the survey and design data typically contained in CAD applications. Collection of most GIS data occurs via handheld GPS devices or is generated from aerial imagery, while survey-grade GPS or survey methods with higher accuracy are used to collect data associated with CAD software. The information in the following sections focuses on aerial imagery and applies to the imagery itself and any data generated from that imagery.

Online map services, such as Google Earth and Maps, Bing Maps, and Open Street Map, are very helpful in conducting virtual site visits and assisting in familiarizing the modeler with the site of interest. While these are valuable resources, they often do not allow users to download data for use in other software.

4.6.1. Collection Methods

Aerial imagery collection methods include satellite, manned aircraft such as a plane or helicopter, and UAS. All the collection methods can provide imagery sufficient for use as background data useful for 2D modeling. Modelers should understand potential accuracies and limitations of any aerial imagery as they relate to use with 2D models. For instance, if a modeler is attempting to locate a building footprint, it may be necessary to understand what the horizontal accuracy of the image is and how the angle of the image may impact the user's understanding of how the footprint location should be determined.

Aerial imagery, regardless of the platform used, undergoes a process of geometric correction to produce a representation of the earth's surface. This process corrects for camera distortion and topographic relief, requiring a 3D surface model and survey control coordinates. The resulting product is called "orthophotography" or "orthoimagery" and is comprised of a seamless mosaic of individual image frames.

4.6.2. Aerial Imagery and Derivative Product Accuracy

The collection method and processing methods as well as the accuracy of the reference 3D surface model all impact the accuracy of the final image. The quality of the 3D surface model can have significant impacts on the accuracy of the orthoimage (see Figure 4.31). Because of the multiple factors impacting the horizontal accuracy, image pixel size does not automatically equate to image accuracy.
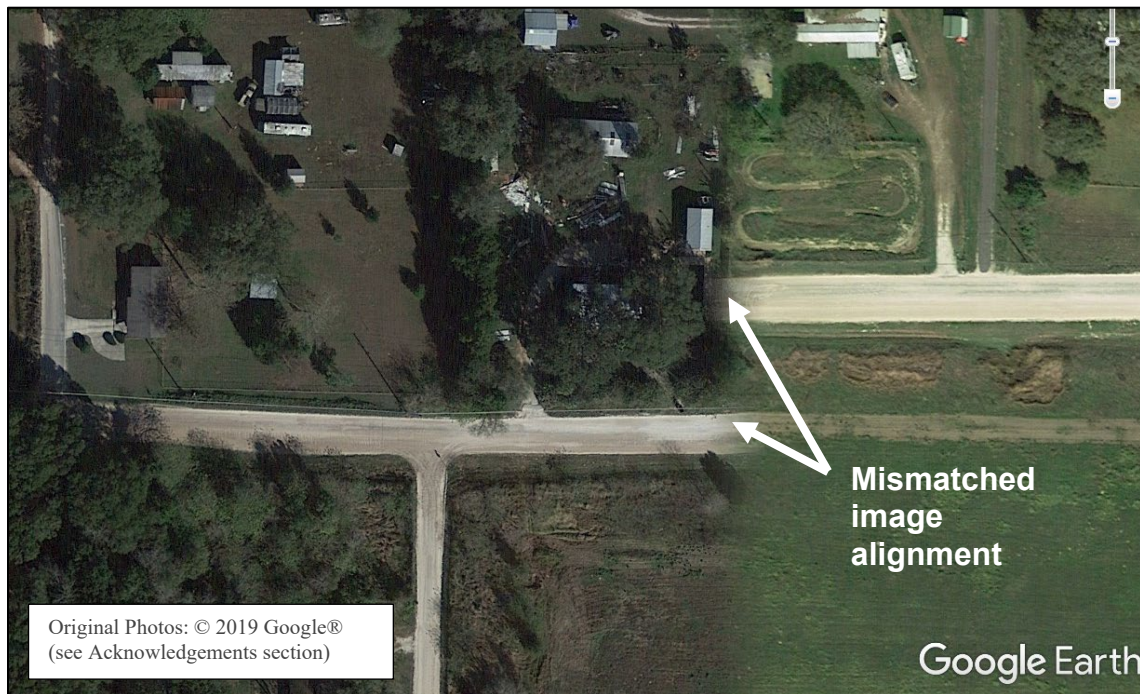
Figure 4.31. Mismatched aerial images as seen in Google Earth.

Important values associated with orthoimagery include image resolution (pixel size) and horizontal accuracy. A variety of resolutions are available from over three feet to as small as one inch. Horizontal accuracies are closely related to pixel size, but also depend on other factors including collection and processing methods and can range from 20 feet to less than one foot, (see Table 4.8). Stereo-compiled planimetrics extracted from aerial imagery references the same positional accuracy standards as orthoimagery. A representative range of pixel sizes and horizontal accuracies are illustrated in Figure 4.32. The highest resolution and up-to-date imagery is typically not necessary for 2D model analysis since imagery is not required. If imagery is being used to determine land use types, the date of the image should be similar to the date of the terrain data so that land use and terrain features match.

Table 4.8 Example Horizontal Accuracy for Digital Orthoimagery.

Common Orthoimagery Pixel Sizes (in)	New Standard RMSE x,y (in)	at the 95% Confidence Level (in)
0.2	0.5	1.2
1.0	2.0	4.8
3.0	5.9	14.4
12.0	23.6	57.8
39.4	78.7	192.7
Source: https://www.asprs.org/wp-content/uploads/2015/01/ASPRS_Positional_Accuracy_Standards_Edition1_Version100_November2014.pdf		
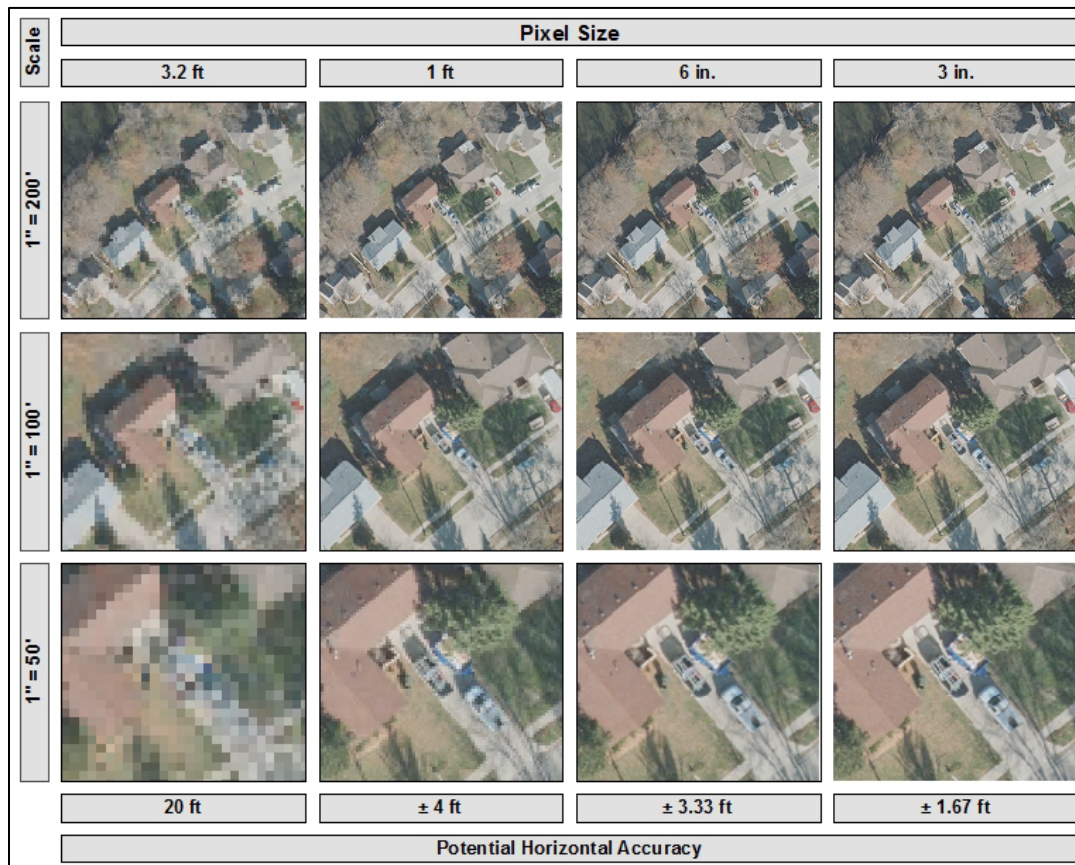
Figure 4.32. Comparison of various imagery resolutions and horizontal accuracy.

It is helpful to understand what imagery is available when determining if additional imagery capture is necessary. Imagery obtained through the United States Department of Agriculture (USDA) for the National Agricultural Imagery Program (NAIP) is available for the continental US and is collected approximately every three years. NAIP imagery is typically 3.2-foot pixels with a horizontal accuracy of approximately 20 feet. However, NAIP imagery is collected during leaf-on conditions, which may obscure the ground and structures in vegetated areas. Imagery with smaller pixel sizes and improved accuracy is often available, though the sources vary greatly.

4.6.3. Formats

Common geospatial software applications, including 2D modeling applications, can work with a variety of image formats. Orthoimagery is a type of raster file, meaning the data is provided in a series of squares (pixels) of a specific size starting at a known location. Two primary types of image formats are typically available: uncompressed imagery and compressed imagery. Standard, uncompressed, images include common formats such as:

- jpeg
- tiff
- png

A wide range of software applications can use orthoimagery in these formats. If these file types are georeferenced, they need an associated world or projection file.

Due to the typically large file sizes of orthoimagery, it is common, and recommended, that aerial image files be in a compressed image format. Imagery compression optimizes the data, making the overall file size smaller and more efficient to store and use in software applications. Common compressed image file formats include:

- ecw
- jpeg2000
- MrSID

Several software applications allow users to access orthoimagery directly from common sources online. Users are not required to save image files since the software can access the imagery and store it. Since accessing imagery online through software applications may slow down 2D model software graphics, it may be more efficient to download imagery for a project area. Downloaded imagery can also be stored and delivered with other model files. While accessing imagery through software applications is acceptable, modelers should determine the date of imagery and verify potential horizontal accuracy, as necessary.

4.7. SEDIMENT TRANSPORT MODELING

Some 2D models are capable of modeling sediment transport using mobile bed hydraulics. Section 8.4 provides an example application and very brief information on the use of sediment transport models. A thorough treatment of sediment transport modeling is out of the scope of this document. Please see Chapter 9 in the HDS-7 document for more information and additional references related to sediment transport concepts. Models with sediment transport capabilities perform three processes at each computational location for each time-step:

1. Calculate flow characteristics
2. Calculate the sediment transport supply
3. Calculate the change in terrain elevation

These computations involve significantly more information and configuration than standard 2D flow computations, including land cover, sediment type, stratigraphy, gradation, bulk density, and upstream supply. The following sections discuss additional data required to run a 2D sediment transport simulation. A good sediment transport model should begin with a good unsteady-flow rigid-bed model. However, most sediment transport modeling problems use a simplified mesh compared to standard rigid-bed models because of the higher computational demands.

Sediment transport equations are subject to significant inherent uncertainty, and it is recommended they be calibrated/verified using field data. Because calibration data is rarely available and difficult to obtain, other methods exist to check sediment transport results. For sediment transport measurement resources see HDS-6 "River Engineering for Highway Encroachments" (FHWA 2001), "Erosion and Sedimentation Manual" (USGS 2006), and "Sediment Transport Measurements: Chapter 5" (Diplas et al. 2008).

4.7.1. Land cover

While a 2D model uses a sediment transport equation to calculate the transport capacity at each location within the mesh, sediment transport equations do not account for the protection that vegetation or revetment may offer to the underlying soil. The modeler should consider how such factors could impact erosion. Section 8.4 discusses methods to handle vegetated and impervious surfaces.

To implement any of these methods, the user is required to determine the location and extents of all types of land cover and should have an understanding of the hydraulic forces that such land cover can withstand. Land use types are most commonly determined from aerial imagery or GIS data (See Section 4.5 for information related to land use data).

4.7.2. Horizontal Sediment Coverage

Sediment types in the channel may differ from the sediment on the channel banks and the floodplain. The user is required to identify the location and extents of each different sediment coverage. The properties of horizontal sediment coverage include stratigraphy, gradation, bulk density (and porosity), and cohesion. The modeler should obtain sufficient data to characterize the soils likely to be exposed to flow by the model simulation.

Horizontal variations in sediment type can be obtained from USDA soil maps (see Figure 4.33), or from local geotechnical explorations.
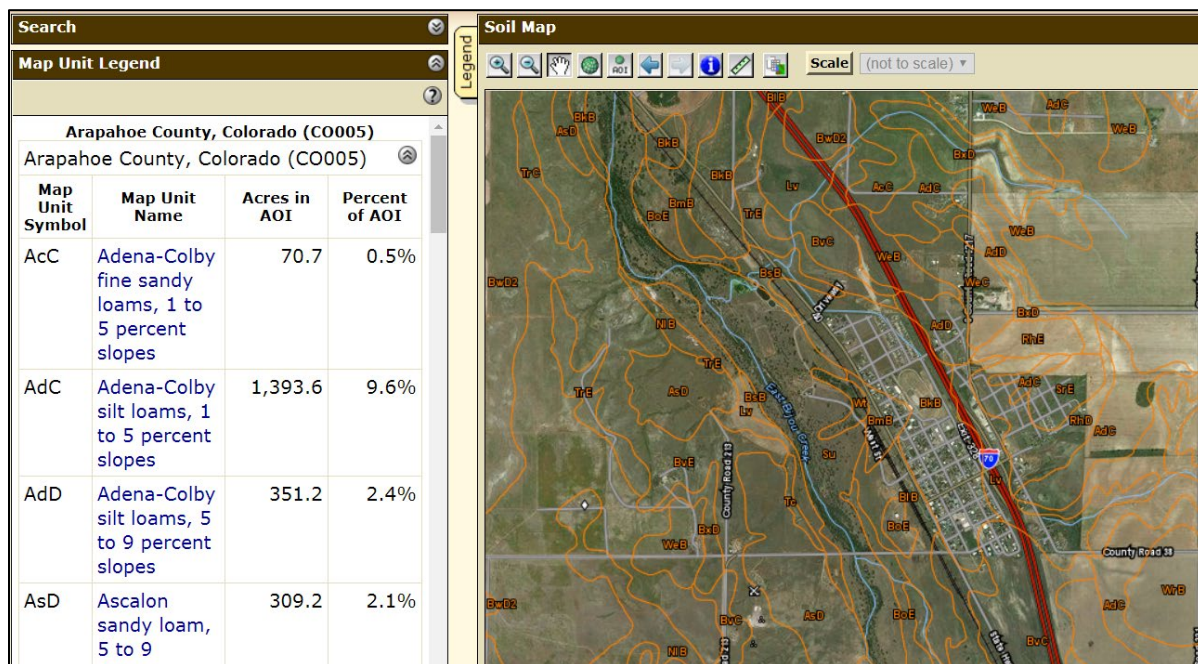

Figure 4.33. USDA soil map.

4.7.3. Stratigraphy

The properties of sediment often change with depth and are needed if the bed is subject to erosion. Each layer of sediment has a different gradation, bulk density, and resistance to erosion. The modeler needs to know the top and bottom elevation of each layer. Geotechnical borings, or other field investigations, can provide this necessary sediment information

4.7.4. Gradation

The hydraulic force required to mobilize sediment varies with particle size. Therefore, more information than the mean particle diameter is necessary. The accuracy and resolution of the sediment gradation data as a significant influence on the usefulness of a sediment transport model. Each sediment layer within each area of similar soil type requires sediment gradation information. Field samples and lab gradation testing are potential sources of gradation information.

4.7.5. Bulk Density

The bulk density of a volume of sediment is crucial to sediment transport models for two reasons. First, the specific gravity, or relative weight of a sediment particle, may directly influence how a sediment transport equation calculates the capacity of the flow to move the particle. Most, but not all, sediment transport equations consider particle density. Secondly, for the model to adjust the terrain elevation, it requires information regarding the volume that a given mass of sediment occupies. Field samples and lab testing are potential sources for bulk density information.

4.7.6. Sediment Supply

The influx of sediment into a model is a necessary component of the boundary condition information for sediment transport analysis. This information is in addition to standard 2D model boundary condition parameters. Sediment data for boundary conditions are typically the most uncertain and difficult data to obtain.

Sediment in natural waterways may be transported either as suspended load or bedload. Suspended load is sediment suspended in flowing water. Bed-load includes material moving near the river or stream bed. The total load, suspended and bedload, need to be accounted for (see HDS-6).

Sediment inflow data should contain a mass flow rate value for each range of sediment particle size entering the system. The format of this data can be a mass flow rate (lb/s), volumetric flow rate (ft^3/s), or a concentration (parts per million (ppm)). The units used for each model and the sediment transport equation are specific to each modeling package. This information is then related to either a time series or a rating table.

Obtaining sediment transport data requires a significant level of effort, time, and expertise. Potential sources of sediment data include some USGS stream gauge data and other databases such as the Brigham Young University sediment transport database. In the absence of field data, the following procedure may be applied:

1. Place the upstream boundary far enough upstream from the area of interest to provide a sediment supply reach.

2. Allow the model to set the upstream boundary condition to the full sediment transport capacity of the flow.
3. Check the results for signs of unreasonable deposition or erosion in the supply reach. Adjust the sediment inflow as needed to resolve these issues.
4. Perform a boundary condition sensitivity analysis by increasing or decreasing the sediment supply significantly. If a change in sediment supply significantly alters the results in the area of interest, move the boundary condition further upstream. In addition to the location of the boundary, varying the sediment distribution may also significantly impact results.
5. Select the location of the boundary sufficiently downstream of a confluence, so that proper mixing of sediment can be achieved.

Several resources exist providing information for obtaining sediment data in rivers and streams for use in engineering analyses including:

- USFS Sampling Surface and Subsurface Particle-Size Distributions in Wadable Gravel- and Cobble-Bed Streams for Analyses in Sediment Transport, Hydraulics, and Streambed Monitoring (USDA 2001).
- USDA Technical Supplement 13A, Guidelines for Sampling Bed Material (USDA 2007).

CHAPTER 5

2D MODEL DEVELOPMENT PROCESS

This chapter includes topics to consider during the 2D model planning and development process. Different models and modeling software may need variations to the standard workflow approach. Modelers should understand the requirements and limitations of their selected model and associated software. Specific model applications may also dictate variations on the standard workflow presented here.

Since 2D models require fewer assumptions than 1D models, the planning and development of a 2D model is often more straightforward than a comparable 1D model. This is especially true when complex hydraulics exist at a project location. As modelers become proficient in working with geospatial data, GIS, and modeling software, developing a 2D model can often be completed more efficiently than a 1D model.

5.1. PLANNING AND WORKFLOW

A successful 2D model analysis requires proper planning before starting a project or when writing a scope of work (see sample 2D scope of work on FHWA hydraulics web page). Understanding the purpose of the analysis and potential uses of the results, assessing the availability of data, and selecting an appropriate model are all essential planning steps. Two-dimensional models are appropriate for any case where a 1D model might typically be applied including riverine and tidal situations in both steady-state and dynamic conditions (see Chapter 8).

Depending on the model application and purpose of the project, specific hydraulic variables are required from the model output. Standard variables useful for hydraulic analysis and design include:

- Velocity magnitude, direction, and distribution
- Depth (and associated WSE)
- Shear stress
- Discharge (split flow, through structures, overtopping)
- Unit discharge
- Stream power

Establishing a typical workflow provides a level of standardization and minimizes the level of effort associated with developing a 2D model. The workflow steps and order presented in Figure 5.1 are generic and can be modified to fit the needs of any project.

As noted in the workflow, it is good practice for the modeler to conduct a site assessment before developing a model to become familiar with the project area topography, assess land use types, determine the detailed geometry of any structures present, and collect photos and notes for later use in developing the model.
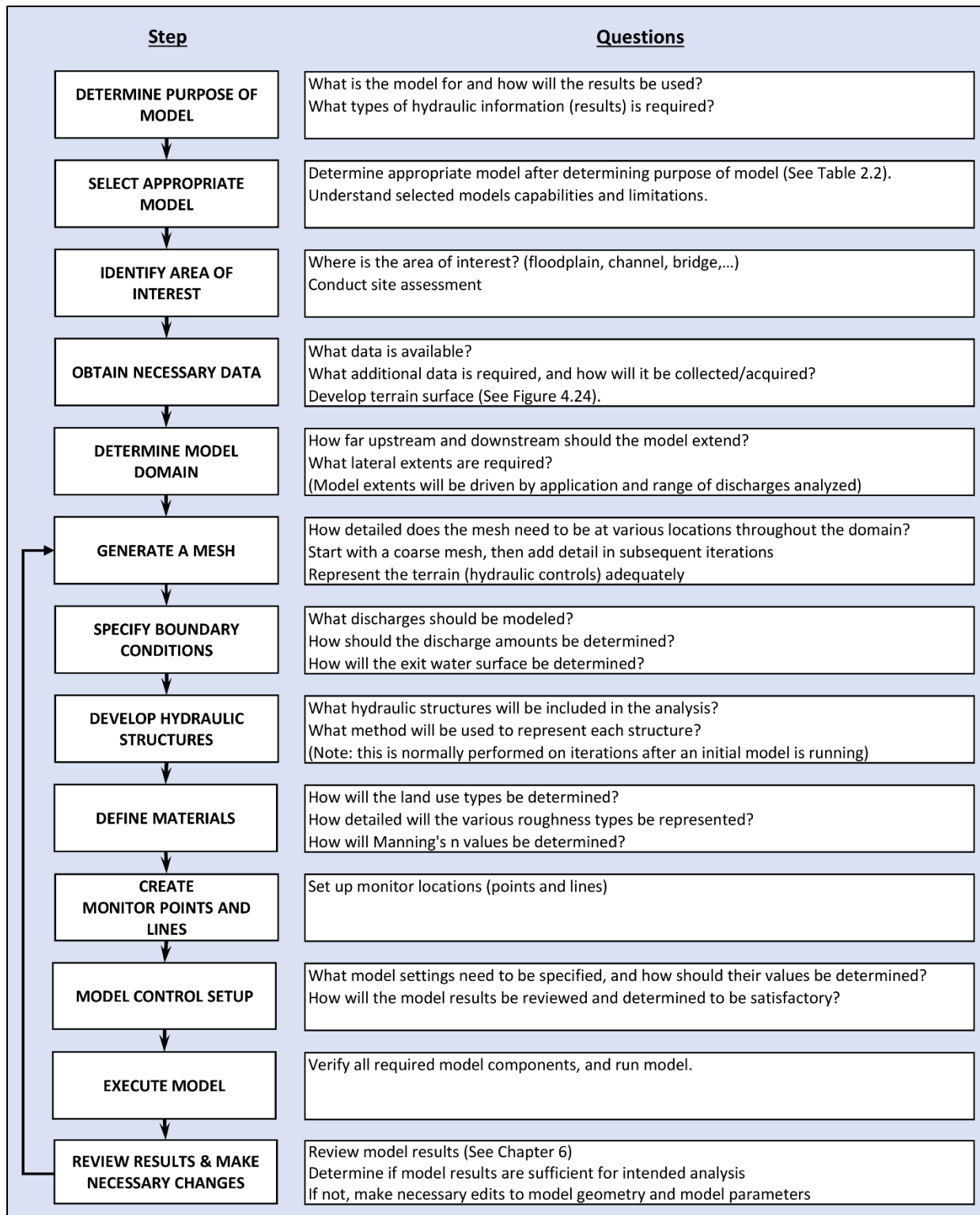
Step	Questions
DETERMINE PURPOSE OF MODEL	What is the model for and how will the results be used? What types of hydraulic information (results) is required?
SELECT APPROPRIATE MODEL	Determine appropriate model after determining purpose of model (See Table 2.2). Understand selected models capabilities and limitations.
IDENTIFY AREA OF INTEREST	Where is the area of interest? (floodplain, channel, bridge,…) Conduct site assessment
OBTAIN NECESSARY DATA	What data is available? What additional data is required, and how will it be collected/acquired? Develop terrain surface (See Figure 4.24).
DETERMINE MODEL DOMAIN	How far upstream and downstream should the model extend? What lateral extents are required? (Model extents will be driven by application and range of discharges analyzed)
GENERATE A MESH	How detailed does the mesh need to be at various locations throughout the domain? Start with a coarse mesh, then add detail in subsequent iterations Represent the terrain (hydraulic controls) adequately
SPECIFY BOUNDARY CONDITIONS	What discharges should be modeled? How should the discharge amounts be determined? How will the exit water surface be determined?
DEVELOP HYDRAULIC STRUCTURES	What hydraulic structures will be included in the analysis? What method will be used to represent each structure? (Note: this is normally performed on iterations after an initial model is running)
DEFINE MATERIALS	How will the land use types be determined? How detailed will the various roughness types be represented? How will Manning's n values be determined?
CREATE MONITOR POINTS AND LINES	Set up monitor locations (points and lines)
MODEL CONTROL SETUP	What model settings need to be specified, and how should their values be determined? How will the model results be reviewed and determined to be satisfactory?
EXECUTE MODEL	Verify all required model components, and run model.
REVIEW RESULTS & MAKE NECESSARY CHANGES	Review model results (See Chapter 6) Determine if model results are sufficient for intended analysis If not, make necessary edits to model geometry and model parameters

Figure 5.1. Typical 2D model development workflow.

5.2

5.2. MODEL DOMAIN

The model domain, illustrated in Figure 5.2 as a yellow line, shows the outer spatial limits of a 2D model. Generally, this area should fully encompass the potential inundated area of the model and extend some distance upstream and downstream from the location of interest. Model boundaries are locations at which flow is introduced or removed from the domain. The location of model limits should be far enough away from the area of interest such that any introduced errors, or deviations from reality, are computationally insignificant in or near the project area.
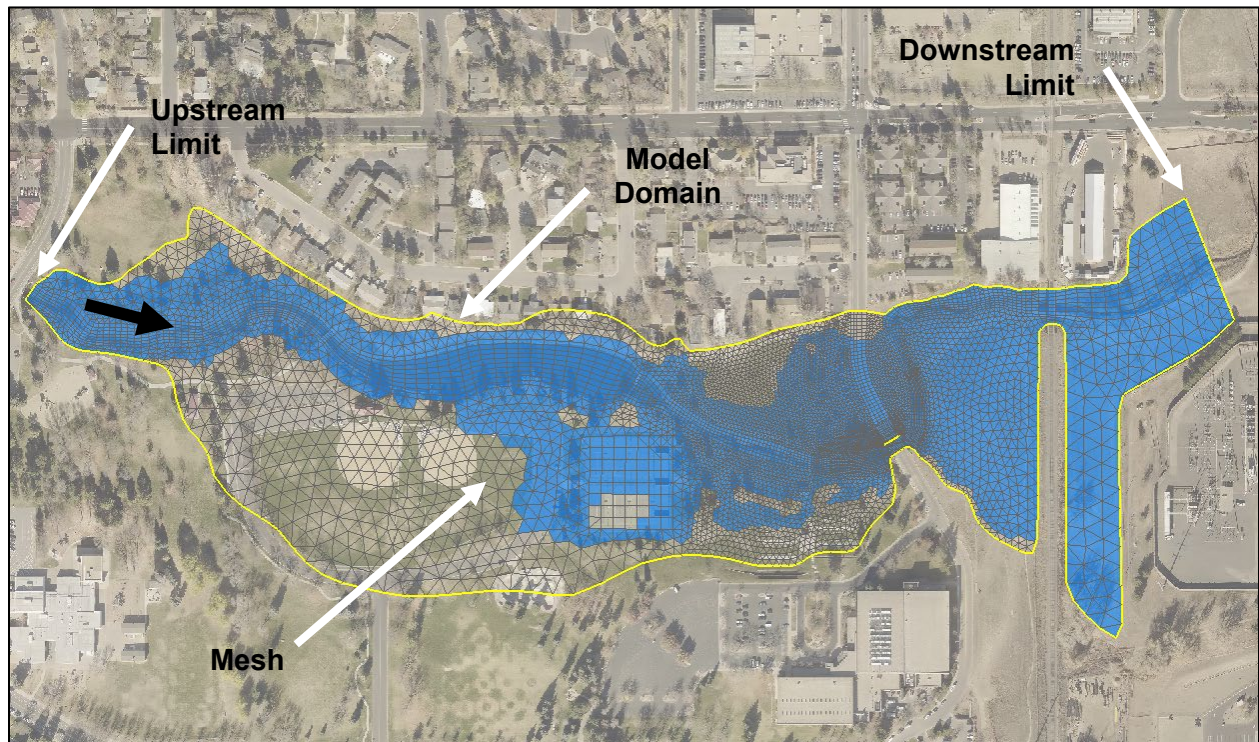
Figure 5.2. Example model domain and limits.

5.2.1. Upstream Limit

The upstream model limit, or limits if the analysis includes multiple inflows, should be located far enough upstream from the area of interest to allow for natural distribution of flow before the area of interest. A good "rule of thumb" for initially locating the upstream limit is about two floodplain widths upstream from the area of interest. This allows the model to work out any potential errors associated with the assumed flow distribution at the boundary so that the hydraulic conditions at the location of interest are natural and physically based. Depending on available data for the project, unique topographic features of the location, and assumptions made, this upstream distance may need to be extended.

The modeler or model dictates the distribution of flow across any inflow boundary. A standard approach to establishing flow distribution is to use conveyance, which distributes the flow across a specified width based on the local channel roughness and available conveyance area. In addition, models commonly assume that flow enters the domain perpendicular to the boundary orientation. This is similar to how a 1D model distributes flow in a cross section.

Once the flow enters the model domain, the model calculates the magnitude and direction of flow. Flow distributes through the domain based on geometry and hydraulic variables. At some point downstream of the inflow boundary, any errors introduced at the boundary are ultimately reduced to zero. The effective distance for this to occur is primarily dependent on channel slope and floodplain width. For steeper, more confined channels, errors are resolved over a short distance, whereas, for flatter channels with wide floodplains, it may take a considerable distance to resolve errors. Also, the lateral extents of flooding near the inflow location may not be known at the upstream limit of the model. If the assumed inflow boundary is wider or narrower than the true extents of flooding, some distance downstream of this inflow the model results are not valid (see Figure 5.3).
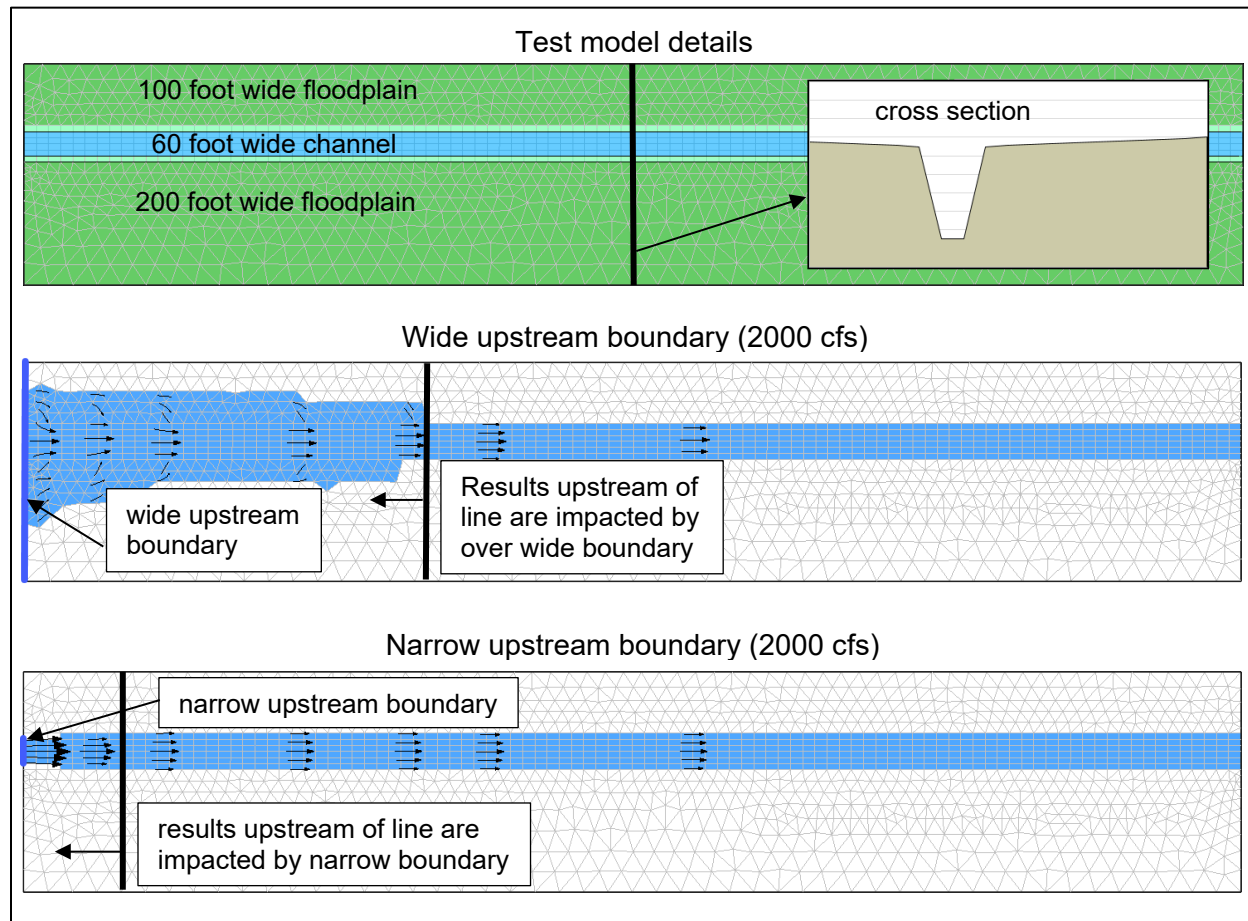
Figure 5.3. Example impacts of upstream boundary condition extents.

Another consideration as to where to place the upstream limit is the extent of the upstream impacts of the project. The "rule of thumb" referenced above may be enough to adequately distribute flow in the model before it reaches the area of interest, but the combination of the project impacts and the valley slope may necessitate a longer distance to the upstream limit. For example, if a river has a very mild slope and a bridge structure of interest is undersized (thereby creating more backwater) the upstream limit should be located far enough upstream to be out of the area impacted by the backwater. In a tidally influenced river situation, the upstream limit should be located far enough upstream to be out of the tidally influenced area.

It is a good practice to locate upstream limits at locations with narrow floodplains and minimal hydraulic complexity. The hydraulics at locations with these characteristics are closer to 1D flow

and therefore, closer to a typically assumed flow distribution. In addition, this approach makes it easier to estimate inflow hydraulics (see Figure 5.4).
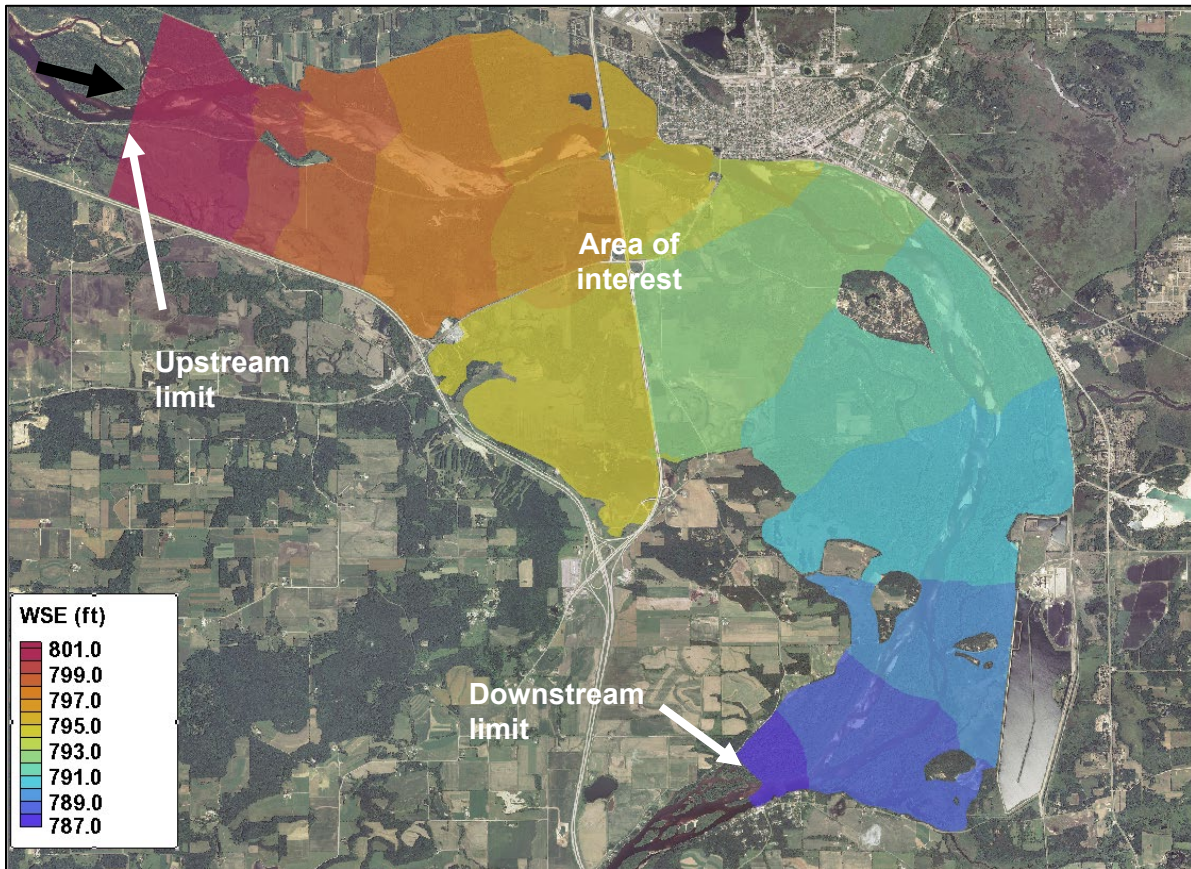

Figure 5.4. Upstream and downstream boundaries located where the floodplain is narrow.

5.2.2. Downstream Limit

The downstream limit can have significant impacts on the accuracy of model results, particularly on river systems with relatively flat slopes, similar to downstream boundaries in 1D models. Where uncertainty exists in the downstream WSE condition, the model should be extended further downstream to reduce the effect of the downstream boundary on the project site. As the model domain extends further from the project area of interest, the accuracy of the terrain data and detailed representation of the terrain is not as critical and approximate data is appropriate if sufficiently far enough away.

Ideal downstream limit locations include sites with known WSEs such as stream gauges, weirs, drop structures, and dams/reservoirs. However, if no measured or controlled value exists at the downstream limit, assumptions are made. As suggested for the upstream limit, a "rule of thumb" of about two floodplain widths can be used to position the downstream limit. Depending on the data available for the project, unique topographic features of the location, and assumptions made, extending the limit downstream may be warranted. If backwater conditions do not exist at the downstream location, the domain can be extended as a "sump" without requiring additional data gathering. As with the upstream limit, it is good practice to locate downstream limits at locations with a narrow floodplain and minimal hydraulic complexity (see Figure 5.4). With any downstream limit, it is good practice to conduct a sensitivity analysis (see Section 6.4) to ensure that variations

in a downstream control do not change model results in the area of interest (see Figure 5.5). The specified downstream water level may be transient, as is the case with a tidal boundary. In this situation, flow may enter the domain from the downstream end.
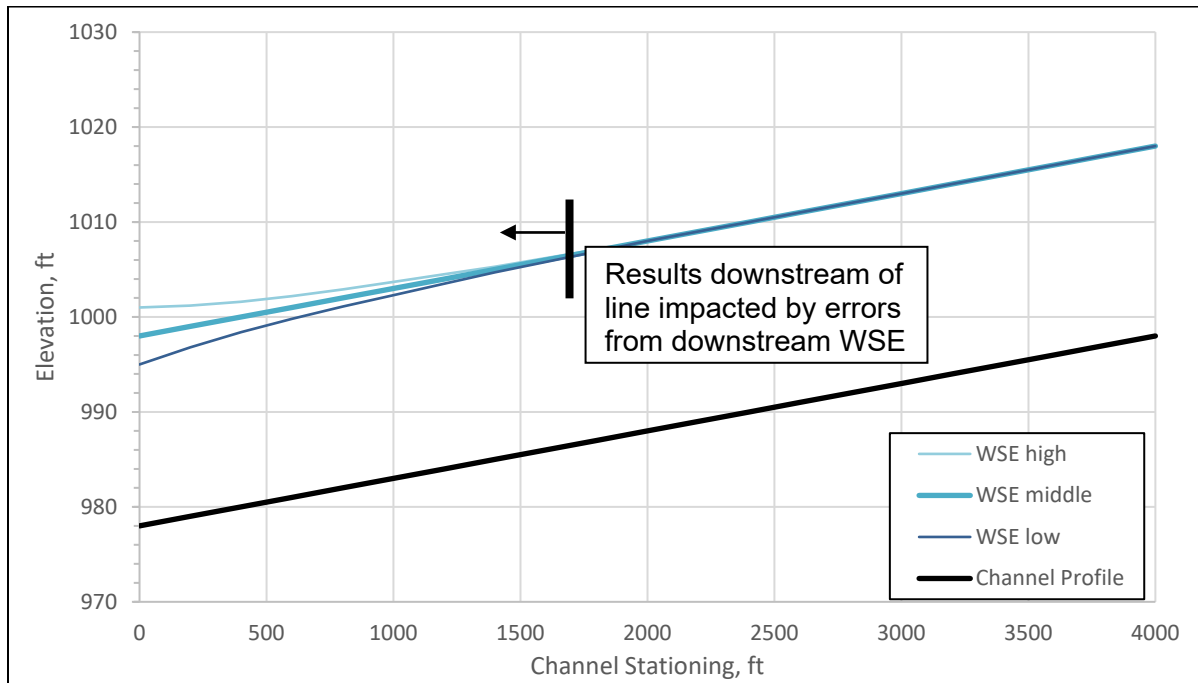

Figure 5.5. Sensitivity analysis of downstream boundary.

5.2.3. Lateral Extents

A 2D model should extend laterally far enough to fully contain all the water or inundated area within the model domain for the discharges to be analyzed. If the lateral extents are not sufficient, the edge of the model acts like a vertical wall, similar to how a 1D model treats the end of a cross section. If the lateral extents are excessively large, then the model runtime could be longer than necessary. It is good practice to extend the lateral extents of the mesh a short distance beyond the inundated area of the largest discharge to be analyzed.

If FEMA 100-yr or 500-yr floodplain boundaries are available, they can help determine the appropriate lateral extents for a model. If the 2D model location is tidal, the lateral extents may include all the land up to a potential maximum high-water surface elevation. In all cases, it is helpful, and good practice, to create and run a coarse model that can provide approximate lateral extents.

Creating a model domain that covers the project location is important. However, some topographic features or modeling requirements may warrant adjustments to the model limits. When floodplain storage is not an important factor, like steady-state models, creating a model encompassing the entire floodplain extents could result in inefficiencies. In the case of a river valley with small side drainages (Figure 5.6), covering those drainages with elements results in increased computation time but does not contribute to hydraulic flow patterns because these areas do not convey flow and would fill up with standing water. In this situation, the domain shown in Figure 5.7 results in the same hydraulics with less computation time.
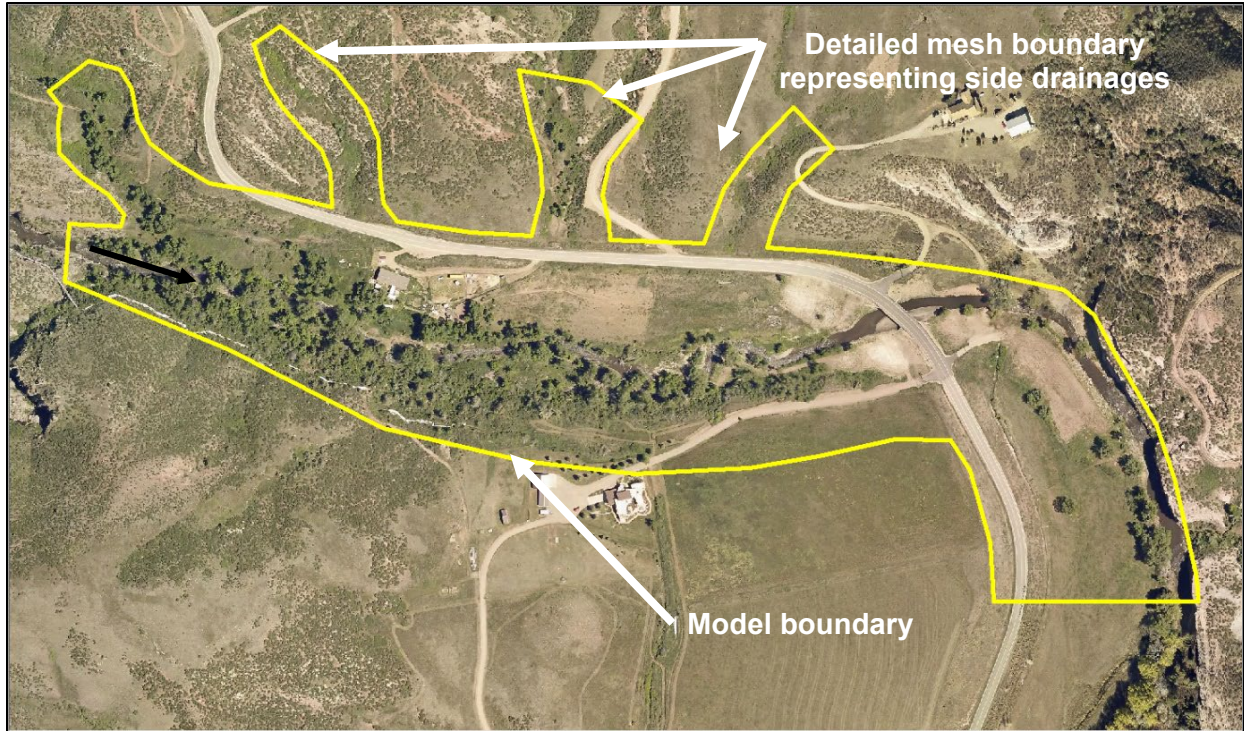
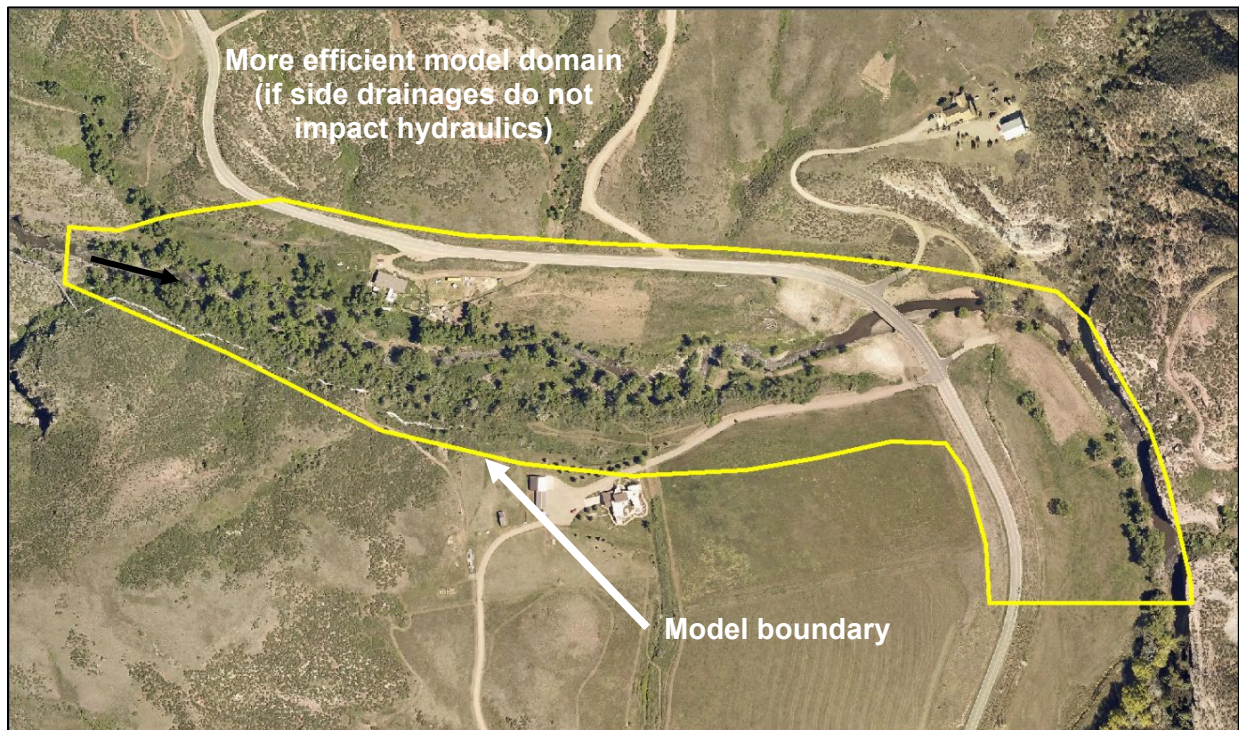Figure 5.6. Model limits including detailed tributary valleys.

Figure 5.7. Model limits excluding detailed tributary valleys.

5.3. MESH DEVELOPMENT

A mesh typically consists of thousands to tens of thousands of elements. Due to the potentially large number of elements, the generation of a mesh has traditionally occupied a significant portion of the time required to perform a 2D simulation. Fortunately, improved tools and utilities to create a mesh are now available. These include traditional CAD and GIS utilities as well as custom software such as SMS and HEC-RAS Mapper. The decision of what tool to use depends on the numerical model to be employed and the format of the data supported by that model.

The fidelity with which the mesh represents the key geometric and geographic features in the domain significantly impacts the validity of the results computed by a numeric model. The omission of a key feature such as a channel that conveys flow can result in higher water levels in the model compared to what would actually occur. Conversely, failing to represent an obstruction, such as an embankment across part of a floodplain, as shown in Figure 5.8, can result in lower water levels or incorrect distribution of flow. The numeric model can only produce hydraulic results as accurate as the terrain surface over which it performs calculations.
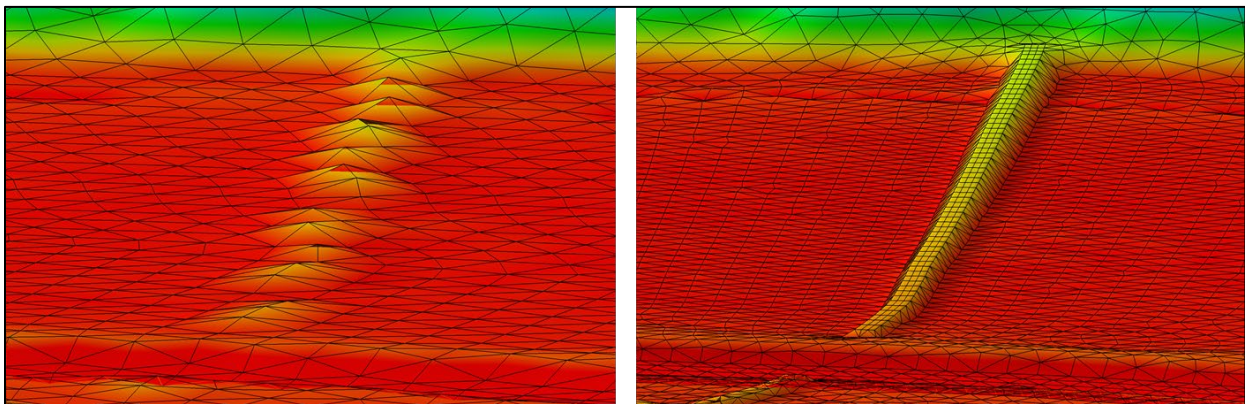

Figure 5.8. Insufficient versus good feature representation of a roadway embankment.

A useful 2D model relies on quality terrain data. The user should ensure the availability of appropriate data before committing to any modeling project. The adage, "garbage in equals garbage out" applies in this situation. Garbage terrain data produces garbage results from a model. When adequate terrain data is available, a few questions still should be addressed during the development of the mesh including:

- How many elements are required (within the area of interest and the entire domain)?
- How should the points representing the surface be distributed to minimize differences between the mesh and the terrain?

The answers to these questions depend on the features and structures included in the domain, the variations in the flow patterns that exist in the domain, and the purpose of the model. A modeler should remember that a flow field can vary greatly in a small area, such as at a contraction through a bridge and around piers. The representing mesh should include enough resolution to capture the significant geometric features that impact conveyance and the variations in the computed flow field and water surface. While sub-grid scale resolution does allow for reduced resolution to represent some geometry, variations in the solution fields requires multiple elements to represent each variation in the flow field. One guiding principle is the realization that three elements are required to capture any variation in flow for even the most simplistic trapezoid channel. Even with sub-grid scale features, the elements should be aligned to the channel with

the element edge along the banks to correctly capture the conveyance. To represent velocity variation across the channel, each side should be subdivided, resulting in a minimum of five elements to represent the flow distribution across the channel. Representing wider channels with variations of features requires additional elements for each feature. The goal of providing a sufficient representation of the flow and velocity distribution guides the determination of how many elements are needed.

Smaller, secondary channels may not need as many elements, but in areas where detailed flow or velocity distribution are needed, multiple elements along the channel bed and banks are recommended. Similarly, an eddy pattern requires at least four elements to represent the variations in the flow field. Once a modeler has identified the features to represent (embankments, channels, structures, etc.), a process can be followed to represent those features in a mesh. This mesh can be used to generate a solution which should then be reviewed to determine if more resolution is required in specific areas.

Because modern tools make the generation of high resolution meshes quite easy, a very common error is for modelers to generate meshes with much higher resolution than is required. A common approach consists of determining a minimum feature size and using that size as a global guideline. For example, a 10-foot spacing may be required to represent the variations of shape in a channel, so a mesh with 10-foot elements could easily be created to represent the entire domain. For a wide floodplain, this could result in hundreds of thousands or even millions of elements. While modern numeric models can compute flow in such a mesh, it requires days of computing time rather than the minutes required by an efficient mesh. In addition to increased computation times, excessive resolution can also result in possible instabilities and slower post-processing times when dealing with the resulting solution data. The best approach is to generate a mesh with enough resolution to represent the important features and no more.

With the 2D models available today, it is good practice to initially develop a coarse mesh by using larger elements to guide the process of generating a detailed mesh for the analysis. A coarse mesh can be developed quickly, runs fast, and provides the modeler with insight into approximate hydraulics and water surface extents that can be used to inform the insertion of addition resolution in appropriate regions of the domain. This refinement process should continue until the solution stops changing.

5.3.1. Mesh or Grid Types

Each numeric model imposes its limits on the type of mesh used as well as the type of elements that make up that mesh. The terms grid and mesh are often used interchangeably, with grid referring to some amount of structure and mesh implying unstructured elements. The following discussion describes the most common types of meshes employed by current 2D models. The list generally starts with the most structured representation and progresses through more flexible options. Figure 5.9 shows example mesh types.
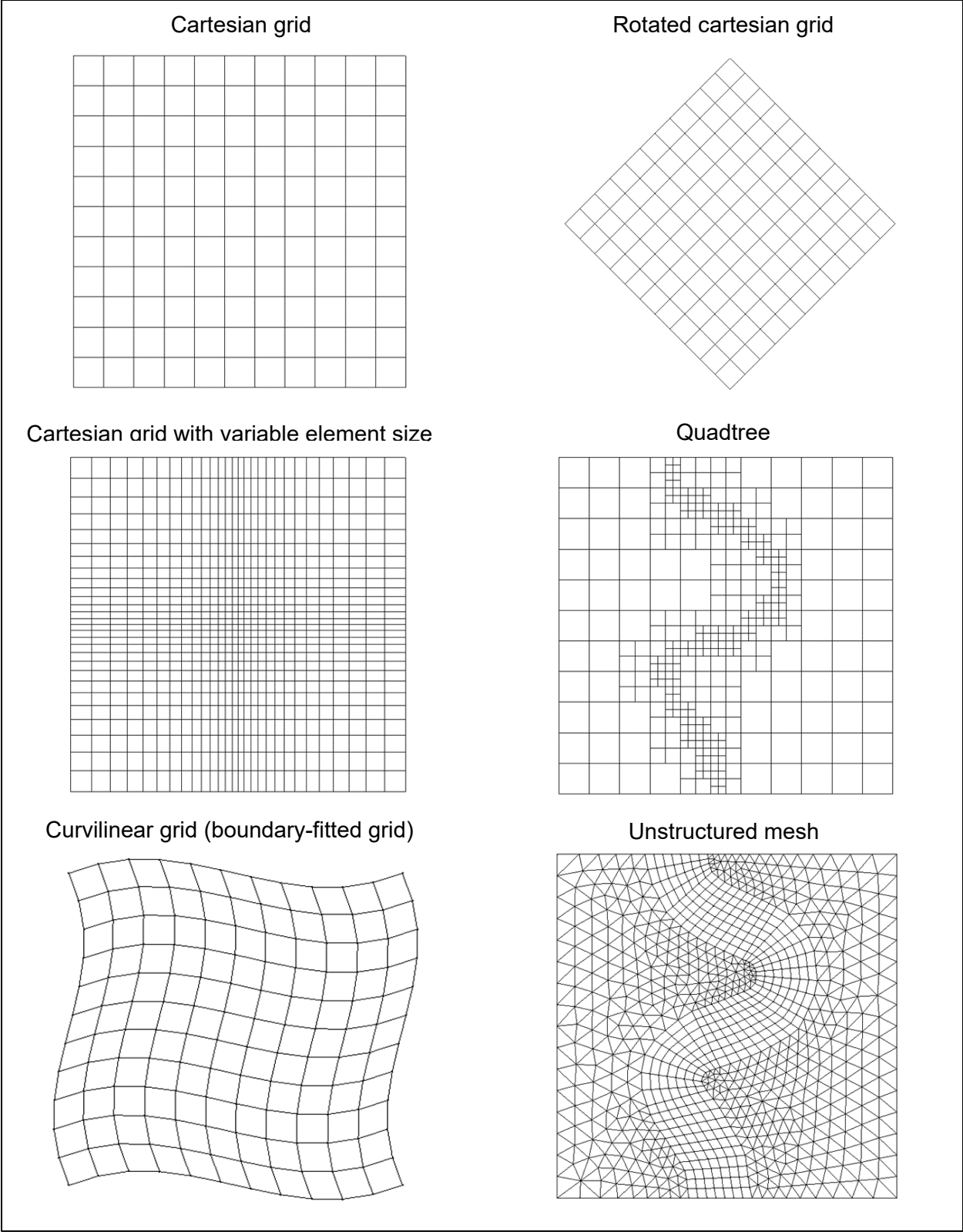
Cartesian grid

Rotated cartesian grid

Cartesian grid with variable element size

Quadtree

Curvilinear grid (boundary-fitted grid)

Unstructured mesh

Figure 5.9. Grid and mesh type examples.

5.10

Cartesian Grid. The most structured format and the most straight-forward to define is a pure Cartesian grid. The definition of this grid consists of an origin, the number of rows and columns, and the element size. In the purest (and restrictive) form, the grid aligns to the designated directions, and all elements have the same edge length in both directions.

This configuration results in element edges that align with the principal directions, and therefore do not represent linear features not aligned with the principal directions. This results in a stair-stepped representation of a given feature.

This definition normally results in smaller elements than the simulation typically needs because all elements are the same size. For example, representing a small channel in the domain requires an element size that is smaller than the channel width. In other portions of the domain, this small element size may result in more density than is needed to represent the natural surface.

Rotated Cartesian Grid. By adding one rotation parameter to the definition of the grid, aligning edges to an orientation other than the principal X and Y directions is possible. However, this formation still only represents features directly in one of two orientations.

Cartesian Grid with variable element size. While still maintaining the simplicity of a grid, variable resolution allows elements to be rectangular rather than square, as well as allowing the column width and row height to vary from one column/row to the next.

Unfortunately, with this formulation, numerical stability often mandates that the ratio of element width to height does not exceed a threshold (often less than 10). Therefore, the flexibility of this approach remains limited. Also, the small row or column dimension still propagates to the edge of the grid rather than just applying to the local area.

Quadtree or Polytree Grids. One final variant of a grid does provide for greater flexibility in resolution allowing for high resolution in areas of complex geometry and rapid variation in the flow to low resolution in areas of slower variations in the geometry/flow. This grid type, referred to as a "quadtree," generally starts with a coarse Cartesian or rotated Cartesian grid. The approach improves accuracy in specific portions of the domain by subdividing an element into smaller elements. The most common approach splits an element into two in both directions creating four smaller elements that fill the region of the original element, hence the name "quad" tree. Other variations allow subdivision of a single element into even smaller elements (i.e., 3x3). These grids generically have the label Polytrees.

By repeating the procedure, very fine resolution is attainable without creating an excessive number of elements. However, the orientation of the element edges remains limited to the orientation of the grid. The stair-step feature representation remains but is minimized because the stair step is smaller in regions of high resolution.

Curvilinear or Boundary Fitted Grid: The next type of grid, referred to as a "curvilinear" or "boundary fitted" grid consists of topologic rectangular elements with four elements connecting at every node. However, the elements are not required to have right angles in the corners and are not limited to any specific orientation.

This approach, while significantly more flexible than the fully structured approach when it comes to representing curved features such as a meandering channel, can become very tricky when trying to maintain consistent rows and columns through complex geometric shapes of varying size such as small channels feeding into larger channels, all of which are nested in a larger floodplain.

<u>Unstructured Mesh.</u> The most flexible type of mesh consists of a random network of triangles and quadrilaterals. General guidelines on element shape should be followed to maximize numerical stability and performance; however, no explicit constraints are imposed on the variation of shape and size. This type of mesh is similar to a faceted surface representing the ground. As facets get smaller (points closer together), the representation is better.

Some models also support the option of a more flexible element shape. One example of this flexibility is the use of adaptive elements. The left side of Figure 5.10 illustrates this methodology, where one edge of an element is connected to two (or more) separate smaller neighboring elements, commonly known as nesting or adaptive resolution. This approach does not require transition elements such as triangles. Thiessen or Voronoi polygons represent another example of a more flexible shape. The right side of Figure 5.10 shows an example of Voronoi polygon elements. General polygon elements allow for more edges. For constrained cases, construction of elements should occur such that the edges between elements are perpendicular bisectors of a line connecting adjacent element centroids.
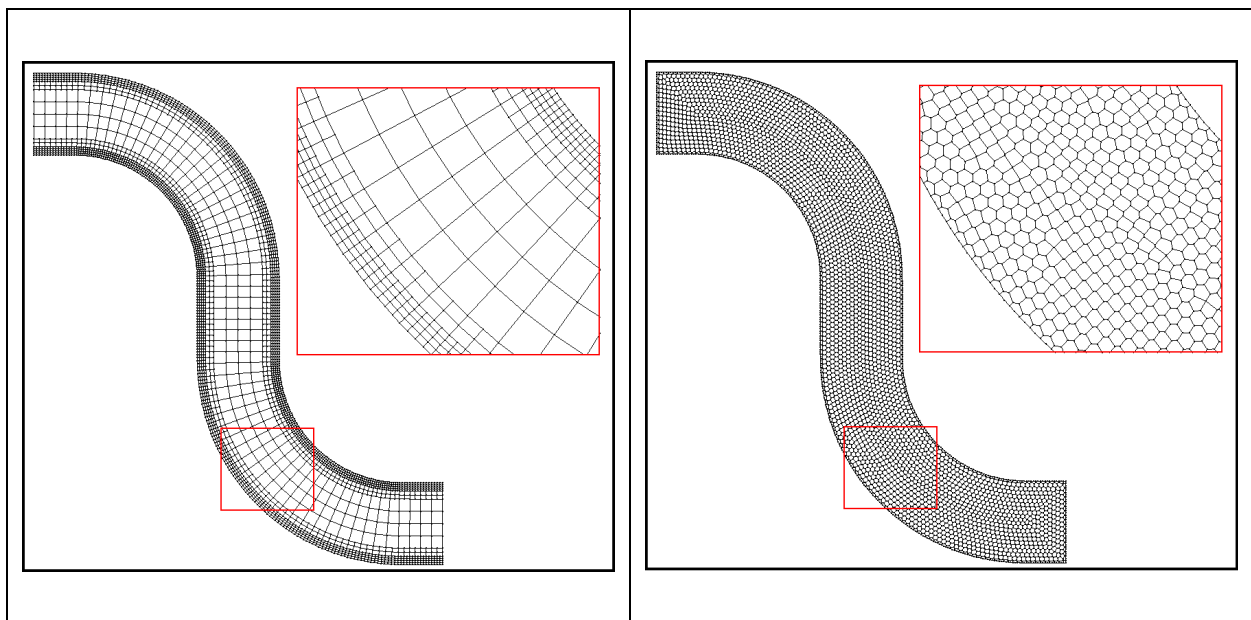
Figure 5.10. Refined (left) and Voronoi (right) grids.

5.3.2. Element Types

The types of elements vary, in the same fashion as the type of mesh varies, from one model to another. A model that performs computations on a Cartesian grid only handles square elements. Similarly, when a model is designed to work only with a specific type of mesh, the shapes should conform to that limitation.

Models that support unstructured meshes impose restrictions on element shape. Some models only support triangular shapes. Others only support quadrilaterals. Others support a mix of both. Others allow N-sided polygonal shapes such as Voronoi polygons. As the number of edges on an element increases, the flexibility to fill a domain with fewer elements increases. For example, when representing a surface with all triangular elements, inserting a single point on the interior of the domain creates two new triangular elements. This results in a typical ratio of twice as many triangular elements as points in the mesh. With meshes consisting of quadrilateral elements, the ratio is closer to 1:1. For example, think of a grid of squares. If there are n rows and m columns

of elements, there are (n+1) rows and (m+1) columns of points. As n and m increase, the ratio of (n*m):((n+1)*(m+1)) approaches 1:1. With Voronoi polygons, the ratio reverses to approximately twice as many points as elements.

Some models only support computations for linear shapes (i.e., a triangle with straight edges) while other models support only quadratic shapes in which endpoints and a mid-side point define each edge. Figure 5.11 provides examples of these two cases. In the quadratic case, the model treats the side as if a quadratic curve is fit through the three points which define the edge. Other models allow elements to have their own order (linear, quadratic, cubic, etc.). Higher-order elements better approximate the geometric surface to be modeled with fewer elements. However, increased computational expense per element offsets some, if not all of the savings of using fewer elements.
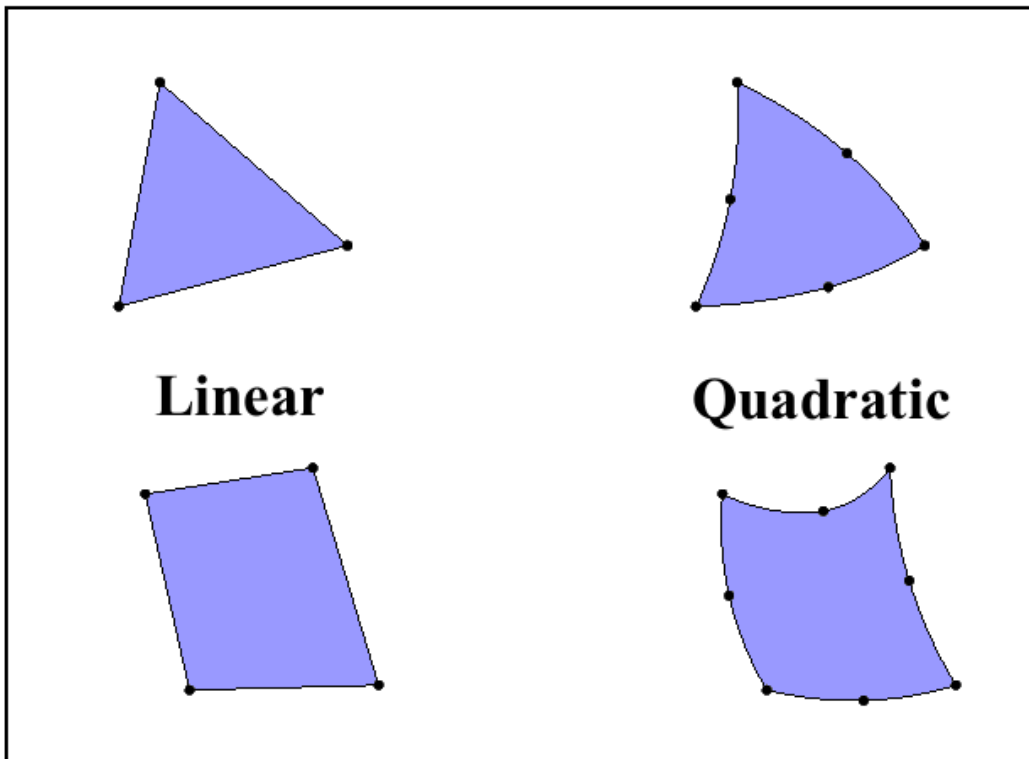
Figure 5.11. Linear and quadratic element examples.

In addition to variability in shape, models also define where the data that defines the mesh is specified. When the elevation is specified at the node locations, the surface shape intuitively varies from one node to the next. This approach works well for finite element solvers that perform calculations at internal points and average the solutions to the nodes. Alternately, the numeric scheme may operate from elevations specified for the element. Visually, this type of representation does not result in a smooth surface, but rather a set of discrete elevations. This discretization is often averaged to result in a smooth surface for visualizations. Some models allow specification of elevations at the element centroid as well as at nodal elevations which can influence the flux across the edge between elements.

5.3.3. Representation of Terrain and Hydraulic Controls

As previously mentioned, the process of mesh generation should consider features that impact the hydraulic flow in the region (see Section 4.2.1). These features should be represented in the mesh for the model to provide a representative depiction of the hydraulic conditions. Creating representative meshes of terrain data is arguably the most important component of developing a 2D model.

Figure 5.12 shows cross sections extracted from three different representations of the same terrain. The brown line represents a lidar survey of a channel section. The red dashed line represents a high-resolution raster (4 foot pixel size) created from the lidar data, and the dashed blue line represents an unstructured mesh with approximately 100 foot cell spacing. The lidar data is a sampling of the actual terrain and should be reviewed by the modeler. When that approximation is sampled to a coarse representation such as the 100 foot mesh, positioning of the remaining points becomes critical to maintain a true representation. While the 100 foot spacing captures general trends in the geometry, it misses local features such as tops of embankments or bottoms of channels. For example, there is a small channel just to the left of the main channel that is lost completely with a 100 foot sampling. If flow in this small side channel is part of the purpose of the study, the resolution should be increased to represent the channel, or the placement of the sampled points should be more carefully controlled to capture the side channel. If the purpose of the study is to resolve the main channel, the 100 foot spacing appears to be quite adequate.
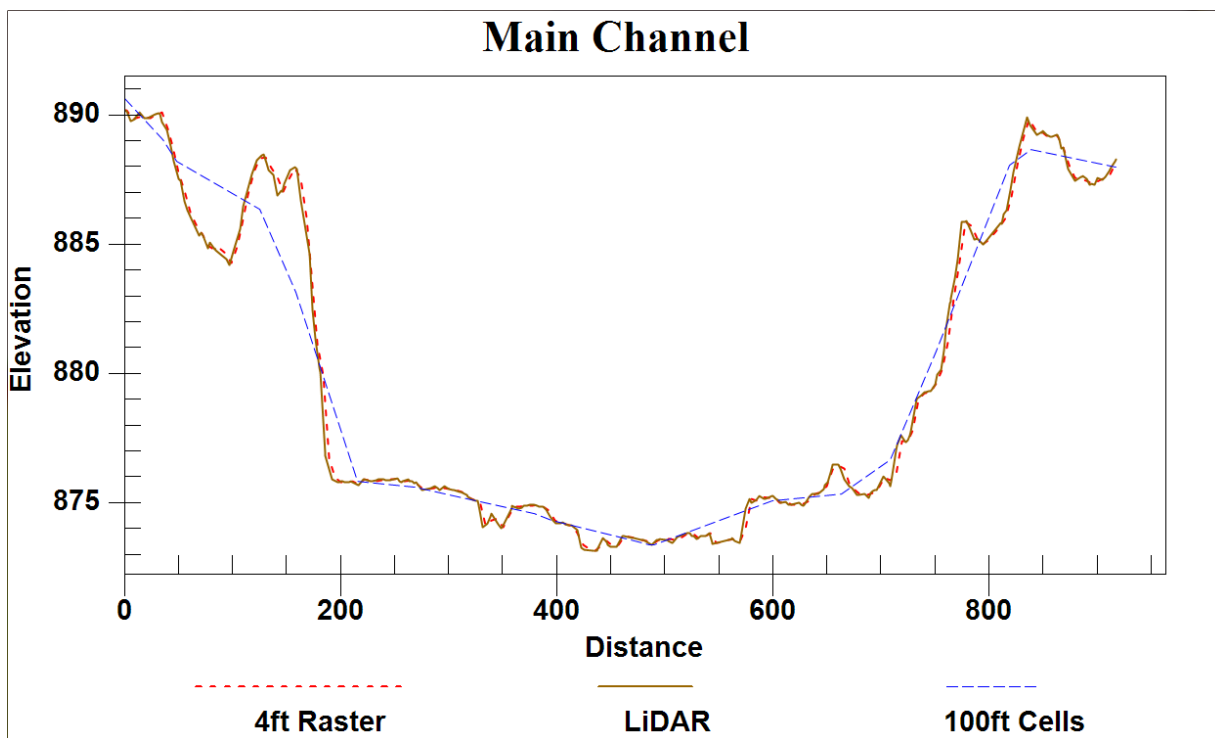

Figure 5.12. Effect of varying resolution on the representation of the terrain surface.

Figure 5.13 illustrates one approach of how a ground surface may be represented by a simplified mesh. The straight red line is the mesh element, and the meandering black line is the ground. In this historical application, the model associated a range of elevations for each node. The size of the range, referred to as storativity (ζ in Figure 5.13), defined a range of elevations over which

the element could convey flow at variable levels. The conveyance decreased from fully effective at the storativity depth above the node to a minimal conveyance (typically about 2%) at the storativity depth below the node. This model allowed minimal flow up to a larger range below the specified elevation (η in Figure 5.13) to prevent numerical shocks when an element dried out.
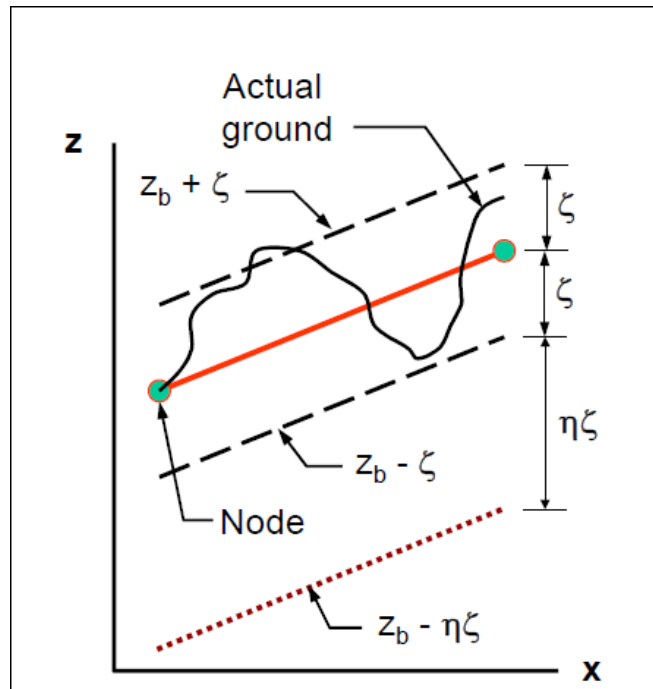

Figure 5.13. Storativity depth assumption (from FESWMS User's Manual).

At any element size, the representation of the surface includes some error. Some models (i.e., HEC-RAS 5.0) store additional information along the edge of an element to account for the approximation that occurs in the mesh representation of the terrain. Instead of storing a single elevation, a minimum elevation is stored, and a curve representing the elevation versus area for an edge or the elevation versus volume for an element is stored. This is commonly referred to as sub-grid scale feature representation. As mentioned, this type of enhancement to the mesh geometry is not new. The FESWMS model (Figure 5.13) called it storativity. Other models have labeled this type of approach "marsh porosity." The additional information stored for the element edges adjusts how much area is available for conveyance or how much volume is available for storage as the model computes the hydrodynamics over the edge or through the element. This additional information impacts the computations when the WSE is above the minimum elevation but below the maximum elevation of the edge or element.

The sub-grid scale approach does not represent the geometric variation of these features. Therefore, the impacts these features have on flow direction are not computed. Studies are needed to validate the impact or improvement these approaches give to hydraulic analysis.

Care should also be taken to ensure that the representation is consistent with the method employed by the model to handle elevations. For example, if elevation values specified at mesh nodes are averaged by the model and applied to element centered values, a single row of nodes at the peak of an embankment or thalweg of a channel may not be detected by the model. Figure 5.14 illustrates two methods of representing an embankment in a mesh. On the left, a single row of points represents the crest of the embankment. On the right, the embankment has three rows of points along the crest. The bottom of Figure 5.14 illustrates a cross section cut through the

embankment. The dotted line represents the actual shape of the embankment, the solid line represents the cross section cut from the mesh, and the shaded blocks represent the element elevations passed to the model.
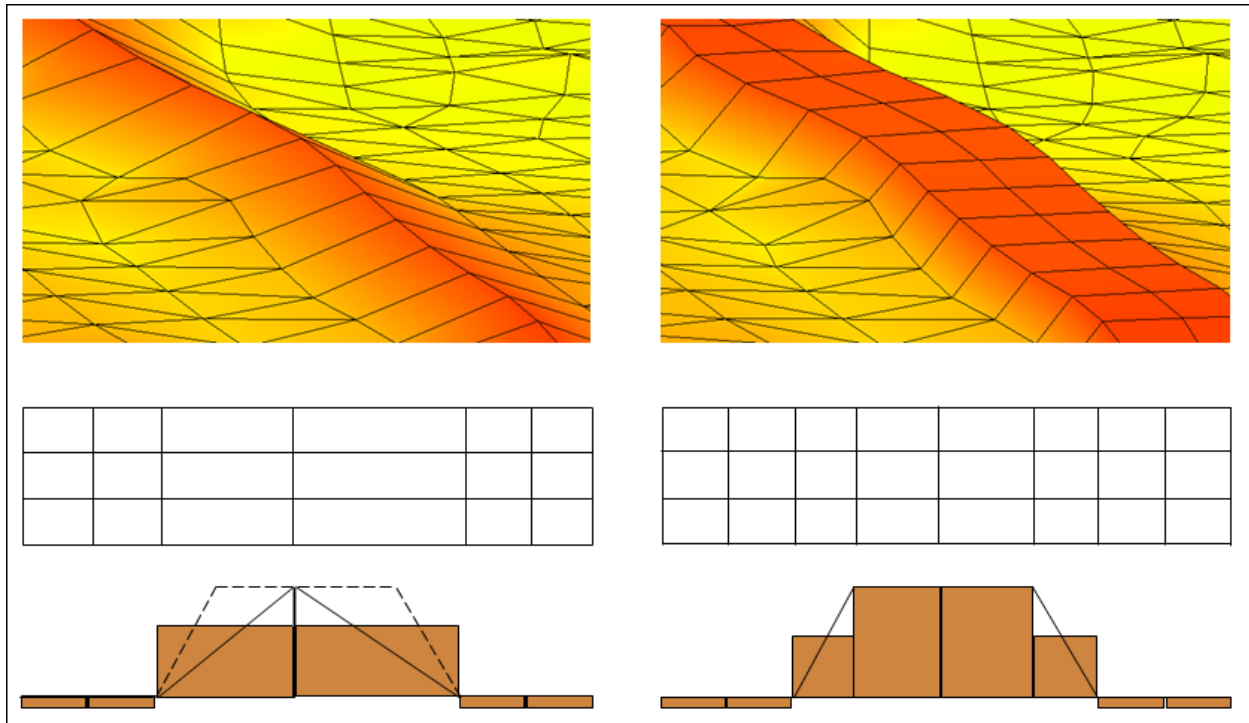

Figure 5.14. Representation of embankment at nodes and elements.

Some models (i.e., TUFLOW and SRH-2D) make use of elevations at the element centers, but also maintain elevations specified at edges for hydraulic barrier definition. The black lines between blocks represent the barrier portion of Figure 5.14. If a model does not store elevations at edges, two rows of nodes (an entire row of elements) should exist along the crest to maintain the correct geometry through the calculations. Similarly, if a model supports sub-grid scale features (such as HEC-RAS 5), care should be taken to ensure that the highest elevations reside along the edge of an element and that the edge does not extend beyond the highest elevation. A modeler should also keep the hydraulic condition around this structure in mind. If a feature is overtopping, additional elements may be necessary to represent the rapidly varying flow.

Recent models may use a more complex spatial variation to this porosity or storativity approach. Sub-grid scale features rely on a high-resolution elevation raster that represents the variability of elevation uniquely for each element and edge. Using this information, rather than increasing the volume, area, or length linearly as the depth increases, these quantities can have unique scales for each element or edge to represent the transition from dry to wet.

As mentioned above, significant hydraulic control features such as channels and embankments should be incorporated into the mesh. This does not necessitate that the resolution increase everywhere, or for that matter at all. It does involve the placement of nodes and elements where the control features exist. Intelligent selection of the resolution to represent these features allows for efficient meshes that adequately represent the terrain.

The following series of meshes illustrate the suggested mesh development workflow. The case includes a bridge crossing in a floodplain approximately 0.5 miles wide. Based on the suggested

guidelines, the domain should extend at least one mile upstream and downstream of the bridge crossing. The channel width of approximately 300 feet would suggest a channel resolution of no more than 40 feet to get 8 sample points across the channel. In addition, there are several smaller relief channels with widths less than 25 feet. Completely resolving these small channels is impractical so they should be approximated using careful resolution placement.

Initially, the modeler defines the domain extents as a polygon. Ideally, inflow and outflow sections cross the floodplain perpendicular to the direction of flow, and lateral extents are at the limits of inundation. Figure 5.15 shows the defined domain for this example over the terrain data.
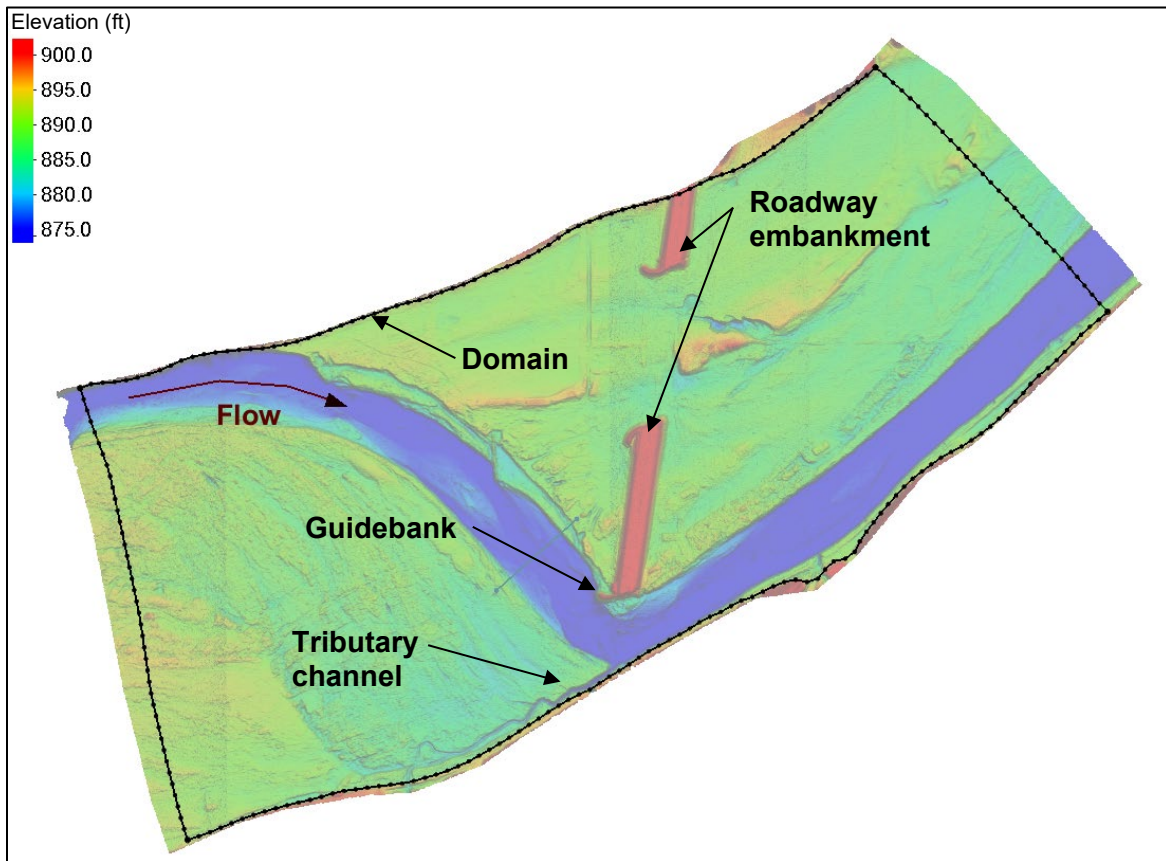
Figure 5.15. Defined model domain extent over terrain data.

At this point, a typical approach would be to generate a mesh of the domain with a constant resolution to determine what terrain features are captured or missed. The number of elements required for a specific node spacing impacts the computing resources required to perform an analysis on the candidate mesh. Figure 5.16 shows the bridge opening for element spacing defined at resolutions of 25 feet, 50 feet, and 100 foot. Table 5.1 provides the number of nodes, elements, and other properties for each mesh. Note the increased fidelity as the resolution increases. Two examples of this are the guidebank and the tributary channel. The guidebank is virtually non-existent at 100-foot spacing, there is a faint representation at 50-foot spacing, and represented at 25-foot spacing. Similarly, the tributary channel does not exist in the 100-foot spacing and is faintly represented in the 50-foot spacing.
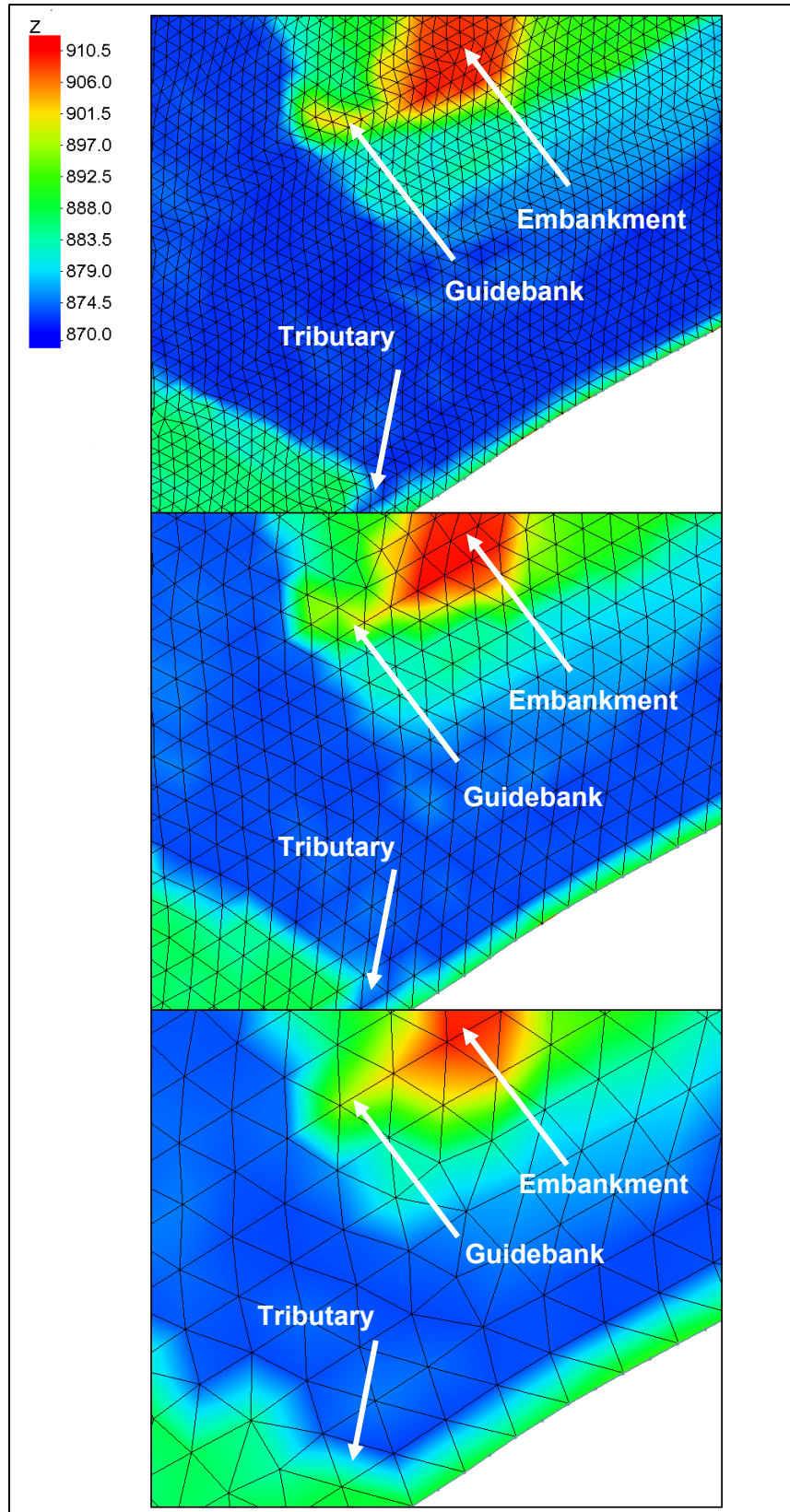
Figure 5.16. Example meshes at 25-foot, 50-foot, and 100-foot resolutions.

Table 5.1. Mesh properties for varying mesh resolutions

Mesh	Properties			
	# Elements	# Nodes	Minimum Resolution	Maximum Resolution
25-foot constant resolution	111,384	56,165	18	30
50-foot constant resolution	26,734	13,604	39	59
100-foot constant resolution	6,752	3,495	74	124
Higher resolution only at hydraulic features	7,154	4,272	45	245

The 25-foot mesh, with over 100,000 elements, has over 16 times as many computations to perform per time step compared to the 100-foot mesh. Smaller elements often necessitate smaller time steps, resulting in more time steps. Therefore, the 25-foot mesh could involve up to 64 times as much computing power (or time) to perform hydraulic analysis compared to the 100-foot mesh. Therefore, a simulation requiring 1 minute to perform with the smaller, low-resolution mesh could take over an hour on the high resolution (25-foot) mesh. The goal of mesh generation is to represent key hydraulic features with as few elements as possible. Sufficient representation is achieved by guiding the mesh generation process to position elements and resolution in key areas.

The top section of Figure 5.17 illustrates the placement of arcs/breaklines in the domain to identify the features with fewer elements. In this example, the channel, embankment and minor channels and ridges are defined. The background resolution is between 100 and 150 feet, which if applied globally, would miss even more features than the 100-foot mesh. However, higher resolution applied to each of the specified locations results in a more efficient representation of the key features. The bottom section of Figure 5.17 illustrates the same zoomed view of the bridge opening in Figure 5.16 for the other meshes. Table 5.1 also includes the statistics for this mesh.

Figure 5.17 also illustrates another common practice to increase the efficiency of 2D models. That practice is to represent features in a channel with long skinny quadrilaterals. These elements are elongated in the direction of flow because the direction and speed of the flow does not change rapidly in that direction. The direction and speed can change rapidly in a short distance across the channel, hence the skinny elements.
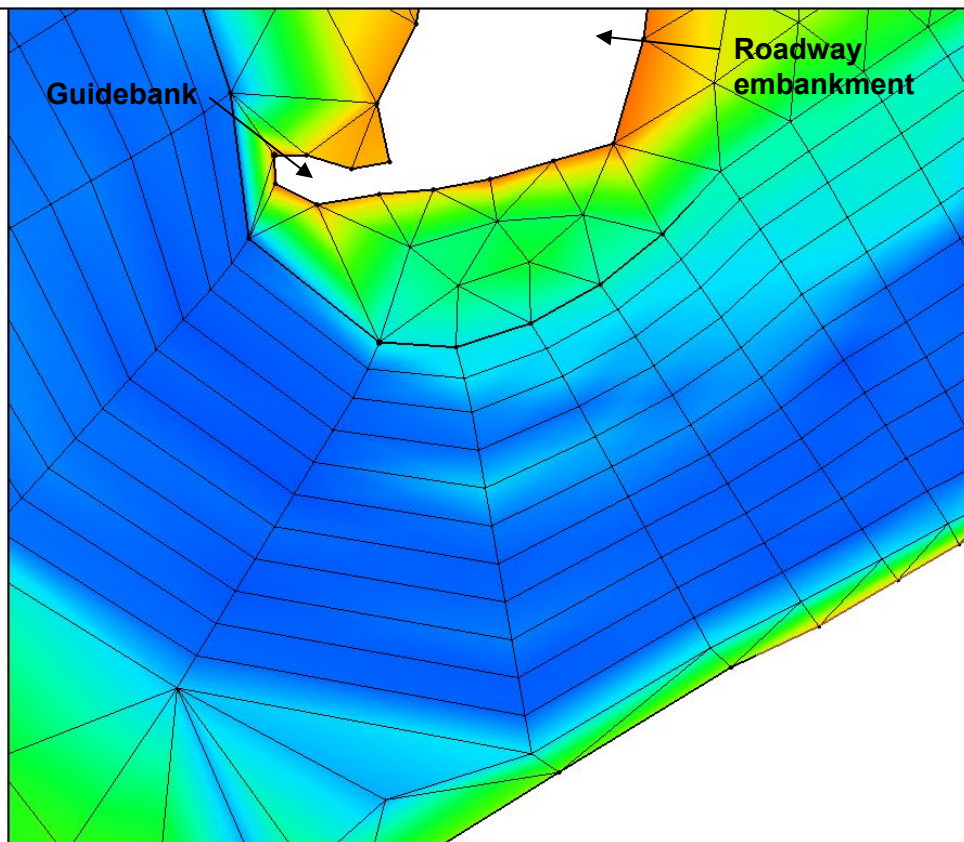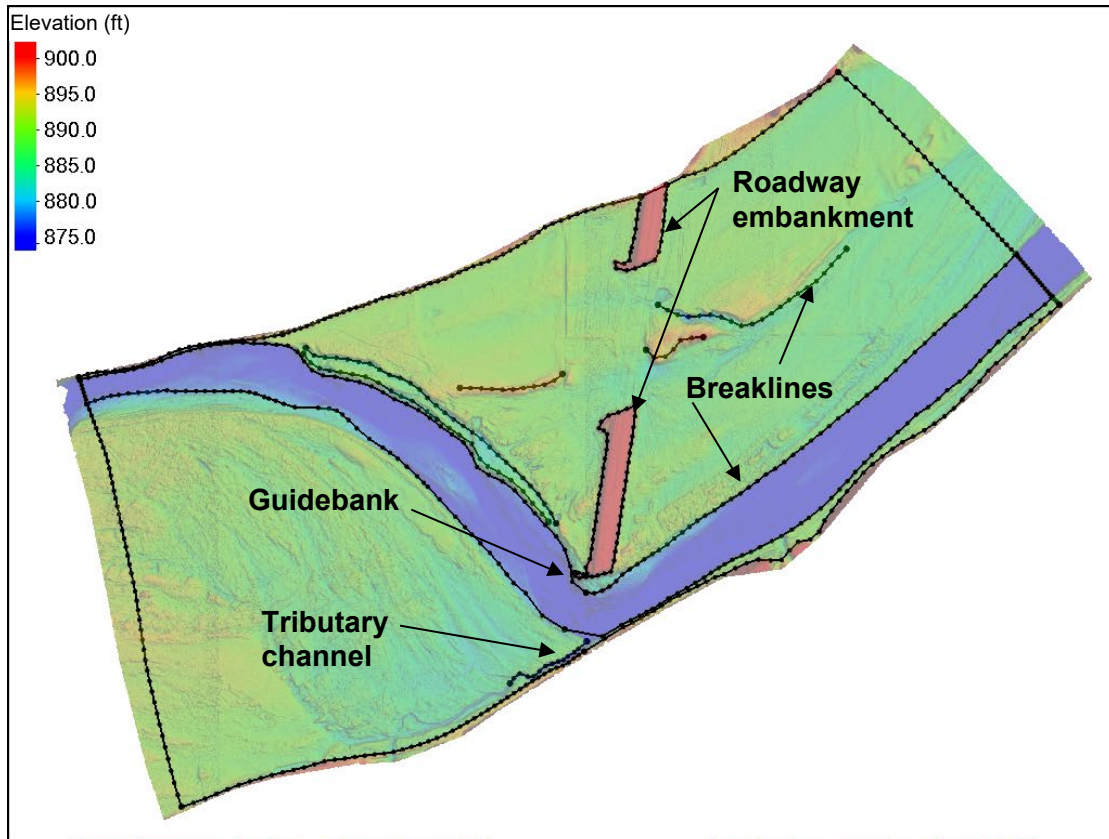
Figure 5.17. Detailed hydraulic features and resulting 2D mesh.

A 2D mesh cannot represent vertical walls except at the edge of the mesh. Similarly, it cannot represent overhanging features because the mesh does not fold back on itself. However, vertical features may be approximated as very steep slopes or as a wall at the edge of the domain. In some models, representing a vertical wall in the interior of the domain is possible by creating a hole in the domain. Steep elements in the mesh used to represent a section that approaches vertical can result in numerical instabilities due to potentially large variations in the solution over a small element. Overhangs should be eliminated by simplifying the mesh. A 3D flow model may be necessary if the simplifications assumed for modeling pressure zones or bridges in 2D are inadequate.

As the resolution of a mesh increases, the size of the elements decreases, and the ability to represent smaller features increases. This leads to questions regarding how many elements are necessary. Generally, modelers should use the smallest number of elements that represent the features of the surface and the flow variations. Tendencies to use too many elements lead to inefficiencies and can lead to numeric instabilities. Some models provide guidelines for the limits on how many elements can be used by that model in a practical situation. If this information is not provided with model documentation, discussion with the modeling community or experimentation can provide this information.

As illustrated above, the mesh generation process may include multiple revisions as the modeler gains familiarity with the site and the chosen model. First pass models may consist of nearly regularly spaced computation points covering the domain. Defining and forcing key features such as the thalweg of a channel or the crest and toes of an embankment into the mesh increases mesh resolution. The numerical results from the model provide additional insight into how the flow varies in the domain. Where flow varies rapidly, the modeler can increase resolution to improve the representation of the flow patterns computed by the model.

5.3.4. Mesh Quality

Mesh quality metrics are usually based on the shape of the element or size of the interior angles in the element (Sarrate 2003). For grid representations (Cartesian grids and Quadtrees), the elements are rectangles and therefore have good shape. However, these same restrictions limit the ability of the grid to represent curved features as well as linear features that don't align with the grid direction. The representation of these features improves as the mesh resolution increases. Unfortunately, this increase in resolution generally results in a decrease in efficiency in the calculation of the numeric solutions.

Curvilinear and unstructured meshes allow for better representation of curved features; however, this flexibility introduces the possibility of poorly shaped elements. Historically, hydraulic models exhibited numeric sensitivity to element shape. Meaning, if elements were not close to equilateral triangles or rectangular quadrangles, the models would diverge, and no solution could be determined. The latest generation of models are not nearly as sensitive to these shapes as older models. However, good practice requires that the modeler review element shapes in the mesh to improve model performance and results of the analysis.

For the example case described section 5.3.3, all the constant resolution meshes have good element shape (near equilateral triangles). Transitions from one resolution to another introduce challenges and potentially poorly formed elements. As resolution increases, the triangular elements tend to get distorted into obtuse shapes, and many triangles can converge at a single location. Indications of poorly shaped elements include extreme interior angles in an element (either small or large), large variations in adjacent element areas, or more than eight elements

converging at a single node. Figure 5.18 shows a portion of the last mesh generated for the above case with potential issues highlighted. The left side of the image displays the mesh generation guidelines along the edge of the channel and embankment. The right side highlights the elements with poor interior angles or rapid resolution transitions. Note that the issues exist at the boundaries of mesh generation zones specified as input for mesh generation. This is where the modeler has specified conflicting resolution requirements. When requirements conflict, poorly shaped element result.

Figure 5.18. Element quality concerns in transition regions.

When generating a mesh, the modeler typically determines the size of the features of interest. The smallest elements in the mesh would be smaller than that minimum size. For example, to represent a pier that is 2 feet wide, the elements cannot be larger than 2 feet. To represent a 10-foot wide berm, the element size should be less than or equal to 10 feet. Using elements larger than the feature often results in that feature being missed in the mesh altogether.

Even with sub-grid scale features stored for the element some features are not explicitly represented. Models produce one solution per element or node, so sub-grid scale features do not produce complex flow patterns caused by those same features being represented explicitly in the mesh. The size of elements often varies from one portion of the domain to another, resulting in small channels far from the area of interest not being represented in the mesh, but with small features near the area of interest included.

Figure 5.19 illustrates an Angle Representation Region (ARR) plot, which is another method of representing the quality of each element schematically. In this methodology, each interior angle of a triangular element is labeled α, β, or χ, where α is the largest angle, β the middle-sized angle and χ the smallest internal angle. Each element in a mesh then can be mapped into a one-sixth portion of an equilateral triangle representing the three angles (Figure 5.19). In this region, points that fall on the dashed line represent elements with a right angle. If the element has ideal shape (an equilateral triangle) it would be represented as a point in the upper right tip of the classification region. As the element quality degrades, the representative point moves towards the bottom of the graph. Thresholds for acceptable standards can be established based on model numerical stability and specified element quality metrics such as Q(αmin) which is based on the largest angle in the element.
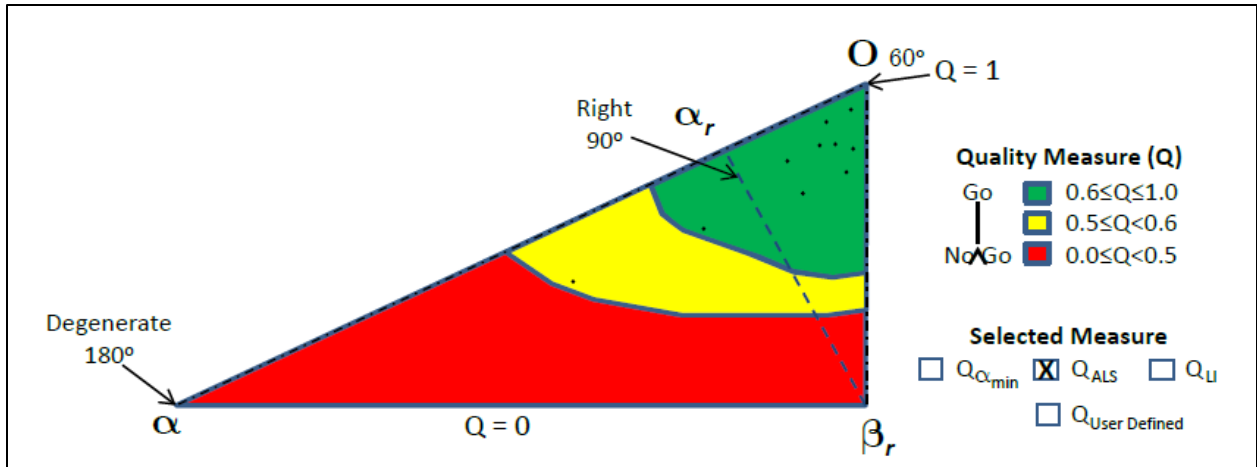
Figure 5.19. Angle Representation Region representing individual element quality.

For the mesh discussed in section 5.3.3, Figure 5.20 shows an ARR plot for the worst 200 elements. As mentioned, each point on the plot corresponds to an element. The skinny triangles near the bottom right side of Figure 5.18 are elements associated with the leftmost points in Figure 5.20. These triangular elements lie between two specified features for mesh construction. Therefore, the element shape is constrained by the user-specified mesh generation zones. By setting the specified resolution associated with the two adjacent features to be more consistent, the shape of these elements could be improved.
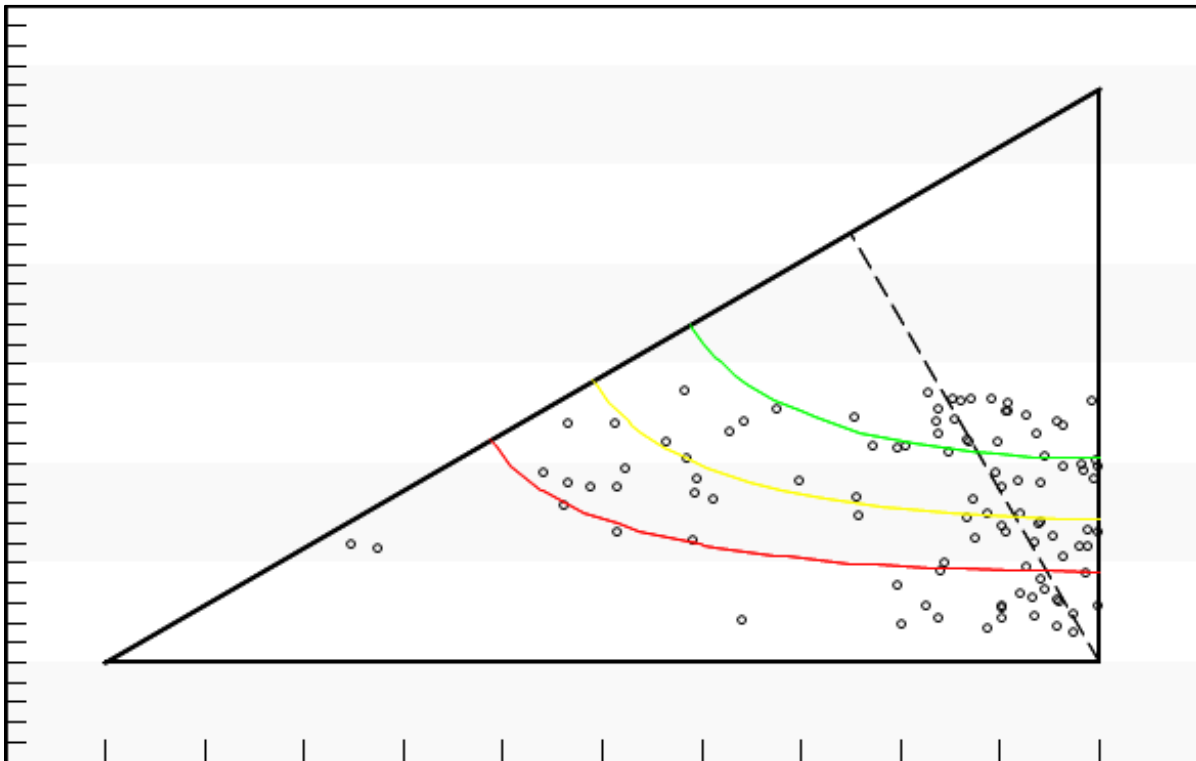
Figure 5.20. Individual element quality plot.

The elevation range of an element also influences numerical stability and results. When a single element spans large elevation ranges, or when the adjacent elements have inconsistent elevation ranges, numerical instabilities can result.

Since all numerical methods use some type of matrix theory, a single instability prevents the creation of any solution. Therefore, reviewing element quality is important. Addressing mesh quality can be approached similarly to how errors, warnings, and notes are dealt with in HEC-RAS 1D modeling. It is standard practice to correct the model to eliminate critical errors and warnings. However, typically not all warnings and notes are significant, and some may not need adjustments to the model.

For 2D models, element and mesh quality reviews using plots and methods similar to those described in this section alert the modeler to potential areas of concern. Elements with severely poor shape metrics should be corrected before a model execution. However, modern models have significant tolerance and can produce results for meshes with low quality. A typical workflow involves generating the mesh, reviewing mesh quality, correcting clusters of obvious issues by changing the constraints used to generate the mesh, and then attempting an analysis. If the model fails, diagnostic messages usually identify offending elements by ID, at which point the mesh can be modified or a new mesh generated to improve element quality in the offending area. If the model generates a solution, reviewing model results can highlight areas where the solution varies too rapidly, oscillates, or appears unreasonable. Generating a new version of the mesh with modified resolution in these areas usually produces a better result. As was mentioned earlier, one approach is to generate successively higher resolution models until the results quit changing.

5.4.   BOUNDARY CONDITIONS

Most modeling scenarios for riverine applications include one or more flow rates (Q) specified at locations where water enters the domain (upstream boundaries) and one or more control water surfaces (head) specified where water typically leaves the domain (downstream boundaries). Other boundary conditions include locations inside the domain where flowrates are computed or influenced by methods other than the standard 2D model solution method. This includes, but is not limited to:

- Sources, sinks, and links
- 1D structures
- Physical processes (meteorology or geology)

Modelers need to understand the fundamentals of data requirements (Section 3.4), sources of data (Section 4.1.1), and where to locate the upstream and downstream boundaries (Section 5.2).

Determining correct upstream and downstream boundary locations and values may involve an iterative approach involving initial data gathering and assessment, results review, and sensitivity analysis, and making adjustments as necessary.

5.4.1. Upstream Boundaries

The definition of an upstream boundary condition for a 2D model begins with a flow rate. Flowrate values are approximations obtained from hydrologic analysis or estimated from field data. Data sources include natural, spatial, and computational uncertainties that limit their precision. Utilization of model calibration and sensitivity techniques should be employed to minimize, quantify, and deal with these uncertainties.

Additionally, 2D models need a description of the distribution of flow across the designated upstream boundary. This spatial distribution of flow is seldom well-understood, and it is impractical to measure. Many models attempt to address this lack of understanding and data by providing multiple methods of distributing the flow. Typical approaches include distributing flow based on the conveyance over each inflow segment, the depth of each inflow segment, or specifying a constant unit flow (q) or constant velocity along the inflow boundary. None of these approximations are likely to match the distribution of a natural river and floodplain situation, but methods based on conveyance generally come close.

The model ultimately computes a natural flow distribution, but the flow distance into the model that this distribution takes to develop is highly dependent on the width and complexity of the floodplain flow and the slope of the channel. As a best practice, locating the upstream model limits where the flow is most constricted can help to minimize the distance it takes to attain a natural flow distribution. Furthermore, it is a best practice to introduce flow along a boundary width that is consistent with the flood inundation limits.

5.4.2. Downstream Boundaries

At subcritical downstream boundaries, where flow leaves the domain, the boundary condition requires the specification of a control WSE or depth. As with flow, most models need a depth at each point along the boundary. Since natural variations in water levels across a flood plain are difficult to measure, an approximation of a single value is standard practice. For historic flowrates, observed field data may supply this value. However, field data does not exist for most design flows. In these cases, Manning's equation can be used to compute an approximate value. To use this approach, the modeler should estimate a composite Manning roughness coefficient and friction slope (commonly referred to as normal depth.) For model domains that include a body of water at their downstream end, still pool levels can be used.

5.4.3. Sources, Sinks, and Links

Sources and sinks, both internal boundaries, can be used to simulate discharge entering or leaving the model from an internal location within the mesh. Sources represent flow entering the domain just like an inlet on the boundary, but in this case, the water appears at an internal node, element, or along edges. How the water enters the domain varies from one model to another. Sources can represent an outfall of a storm drain, detention pond, pump station or similar situations. Sinks can remove or send water out of the model domain. As with sources, this can be at a point, in an element or across an edge, depending on model capabilities. A sink can represent several hydraulic situations, including water flowing into structures like wells, storm drains, or culverts.

A link represents a combination of these two conditions. Water is removed from the domain at a location (the sink) and reenters the domain at another location (the source). This feature can be used to simulate flow between two unconnected elements or sections of mesh. Normally the user also specifies the time lag between water leaving the sink and entering the source. Sources, sinks, and links can represent a constant flow rate, a time series, or a rating curve. Some models allow the 2D domain to interact with a full network of these type of structures to simulate storm drain systems.

5.4.4. Structures

Incorporating existing or proposed structures into a 2D model poses additional opportunities and challenges to the modeler. Most models treat structures in the domain as a type of boundary

condition, but for this document, they are addressed independently in Section 5.5. As Figure 5.1 illustrates, the process of building a model is iterative. Best practice suggests that a model be built first with as simplistic a representation as possible. This means that bridge structures would be initially modeled as if they have sufficient freeboard to be out of the water. Once the hydraulics of the site are represented, structures can be added as described in the 5.5.

5.4.5. Physical Processes (Meteorology and Geology)

Some models incorporate the ability to represent other physical processes that impact the hydraulics or hydrology of a scenario. Each model is different and supports simulation of the processes deemed important for the application of the model. These processes could include:

- Rainfall and evaporation
- Infiltration and exfiltration (interaction with groundwater)
- Wind or atmospheric pressure
- Tides and waves

Simulation of hydrologic processes normally includes the designation of regions (polygons) in which the processes occur. This can be visualized as a boundary condition applied over an area rather than along a line or at a point. The model normally compiles the spatial inflow or outflow into a total inflow or outflow by multiplying the rate by the area of the region.

Inclusion of meteorological factors can be significant when simulating storm surge generated by wind and pressure fields from tropical depressions. These parameters are typically specified by the user and provided to the model in the form of data sets with values at uniform mesh locations or on each element of the mesh.

In coastal regions, hydraulic and hydrodynamic forces should be combined with coastal processes such as tidal generation and propagation, wave stresses created as waves break on the coast, and wind setup of the water surface. These topics are beyond the scope of this reference document.

5.4.6. Supercritical boundary conditions

Many models support options to specify boundary conditions at locations with supercritical flow. In this situation, inflow boundary specification is more complicated because it requires a control depth in addition to the flow rate, direction, and distribution. Supercritical outflow boundaries need less data due to the fact that in supercritical flow regimes no upstream influence exists. Therefore, supercritical outflow locations need no additional data beyond the specification that supercritical flow exists.

Using supercritical boundary conditions is typically avoided due to the increased possibility of rapidly varied flow conditions that exist in and around supercritical flow zones. These variable conditions increase the difficulty of specifying boundary conditions that match existing physical conditions. Extending the model domain and allowing the numerics to handle the transition to supercritical flow results in a better representation of the natural flow. Erroneous specifications may result in numerical instabilities as the internal solution attempts to correct the unnatural specifications.

5.5. HYDRAULIC STRUCTURE FEATURES

One of the major benefits 2D modeling provides to the practitioner is an improved characterization of flow patterns through bridges, near culverts, and over weirs when compared to 1D modeling and other simpler analytical methods. Most structures consist of complex shapes with abrupt features. They also influence flow patterns and can induce complex and rapidly varying flow conditions. Complex flow conditions include flow regime transitions, rapid spatial changes in flow direction, formation of eddy flow patterns, flow splits, pressure flow, orifice flow, weir flow, and others.

While representing natural features involves care when building a mesh, hydraulic structures present unique challenges and may include vertical walls, enclosed structures such as culverts and bridges, and complex horizontal and vertical structures such as complex piers. Two-dimensional meshes cannot directly represent some of these complexities explicitly within the mesh geometry, but methods are available in some models for incorporating structural features.

Typically, three methods are employed by the various models to represent hydraulic structures. These include:

1. Building the structure into the mesh (explicit representation).
2. Characterizing the effects of the structure as an attribute of the element that contains the structure (examples of this include additional drag added by an obstruction inside an element, or sub-grid scale feature attributes).
3. Representing the structure with a separate (normally 1D) model, or calculation, and coupling that model, or results, with the 2D calculations.

Representing structures explicitly with 2D mesh elements (method 1) is preferable, when practical. A common example is overtopping flow at an approach roadway embankment near a bridge. Although the profile along the top of the embankment could be represented as a 1D weir structure (method 3) within a 2D model domain, representing the embankment geometry with multiple mesh elements, laterally and longitudinally, results in a better representation of the geometric feature within the model with more realistic results. Regardless of the type of structure, 1D equations assume an average velocity and uniform flow distribution and neglect inertial forces (momentum); whereas, 2D model computations compute the actual flow distribution and account for inertial forces.

Support of these approaches varies greatly from model to model. Some do not include specific features such as 2D representation of pressure flow, modeling of flow regime transitions, or inclusion of specific structure types. Therefore, the modeler should select a model with features and capabilities that are best suited for their applications. Section 2.4 presents several models and their capabilities, along with a discussion of recommended model capabilities for highway encroachment evaluations.

The following sections describe methods for including hydraulic structures in a 2D model together with a description of how to represent each structure type using the three methods listed above.

5.5.1. Bridges

Representing a bridge crossing in a 2D model includes the ground surface under the bridge, abutments, piers (which can be complex shapes and groups), and a deck (which provides a ceiling and potential pressure flow, as well as a potential overtopping crest elevation).

Ground surface under the bridge. The 2D model representation of flow through a bridge opening applies to the ground under the bridge. The modeler incorporates this surface into the mesh for representation of flow through the opening, as shown in Figure 5.21. The 2D solution naturally handles the contraction, expansion, and distribution of flow through that portion of the mesh. The responsibility for using enough elements to capture these effects rests with the modeler. If the simulated scenario does not include the situation where the water comes into contact with the bottom of the deck or overtopping adjacent abutments, this representation of the area under the bridge suffices to simulate the flow through the bridge. Typically, element spacing in the area of a bridge is selected so that several elements are located under the bridge in both directions, laterally as well as upstream to downstream.
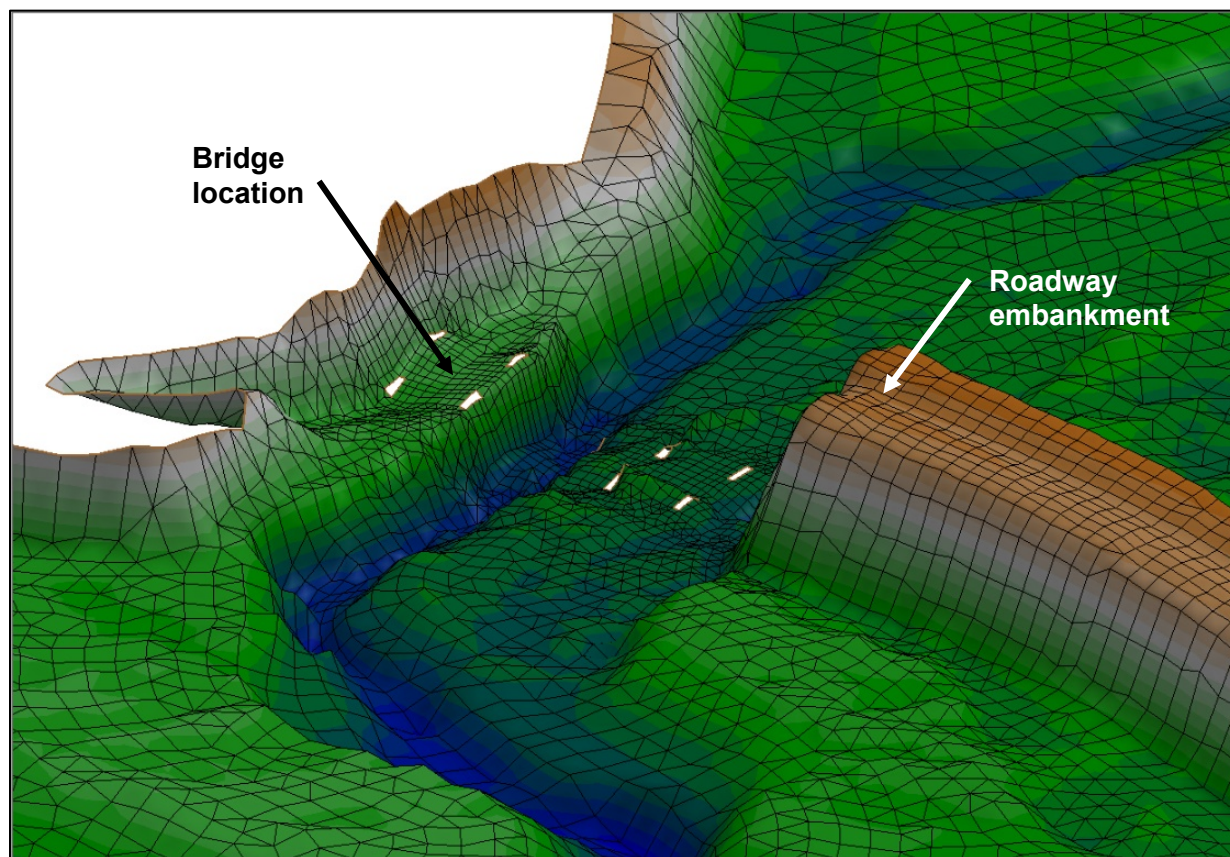
Figure 5.21. Oblique view of mesh geometry representing the ground at a bridge.

Bridge abutments: The abutments adjacent to a bridge opening are typically represented as elements in the mesh covering the inundated area, as illustrated in Figure 5.21. As with the area under the bridge, the 2D flow computations include the flow around the abutments. If the water level in the simulation exceeds the approach elevations, these regions should also be covered in elements and incorporated into the mesh.

Bridge piers: Piers supporting a bridge can be included in a simulation by incorporating them into the mesh, typically as holes (as shown in Figure 5.21), or by accounting for the drag force within the element containing the pier. If the element size under the bridge is smaller than the pier dimension, the first option is viable. If the element size is larger than the pier dimensions, the second option, treating the pier as a detail to the containing element makes sense. If defining flow patterns around piers or defining the flow distribution through the bridge opening, including splits

around piers, mesh resolution should be increased to allow for incorporation of the piers into the mesh definition. Because of the 2D hydraulics assumptions outlined in Chapter 2 and the 3D nature of physical flow around piers, resolving the hydraulics beyond the point of two to three elements across a pier is not justified, nor recommended.

Some models do not allow holes in the mesh. In that case, if the pier is to be represented in the mesh, the mesh resolution around the pier should be increased, and the elevation of nodes inside the pier increased to create a representation of the obstruction to flow. It is important for the modeler to understand how piers should be represented to provide detailed and accurate hydraulic results for the specific model selected for analysis.

Bridge decks: The bridge deck should be accounted for when the water level in a model reaches the elevation of a bridge deck. Because the mesh in this region of the model is already being used to represent flow under the deck, the deck itself should be represented in another fashion. This is traditionally accomplished in one of two ways. The first is to specify additional attributes to the nodes or cells in the area of the bridge. The second is to extract the bridge area from the model and treat it as a separate flow model.

In the first approach, the influence of a bridge deck can be split into two components, pressure flow under the bridge and weir flow over the bridge. Several models allow the specification of a ceiling elevation for each node or element under the bridge. Figure 5.22 illustrates a bridge structure over a mesh. The elevation of the bottom of the deck for each node under that structure would be specified to the model so it can incorporate pressure terms into the partial differential equations for shallow flow. Some models include numeric functionality to simulate the orifice effects at the entrance to the pressure zone.

If flow for the simulation reaches the bottom of the bridge deck, but does not exceed the elevation needed to overtop, this approach suffices to represent the effects of the deck. Often, superelevation of the bridge or the height of the superstructure on the bridge (railing and barriers) result in all overtopping flow occurring around the bridge. In these cases, the model simulation does not need to represent weir flow over the bridge. If weir flow is expected to occur over the bridge deck, the user should also specify the type of weir and the elevation of the crest to allow for 1D computation of flow that leaves the 2D domain at the upstream face of the bridge and reenters the 2D domain at the downstream face.

Some engines do not support features of this type but do support the option to remove the area of the bridge from the model and represent the bridge independently from the 2D simulation (Figure 5.23). Typically, in this situation, the model removes flow at the upstream face of a bridge and redirects that to the bridge model, which is often a simplified 1D representation of the bridge. Flow is returned to the 2D domain at the downstream face.
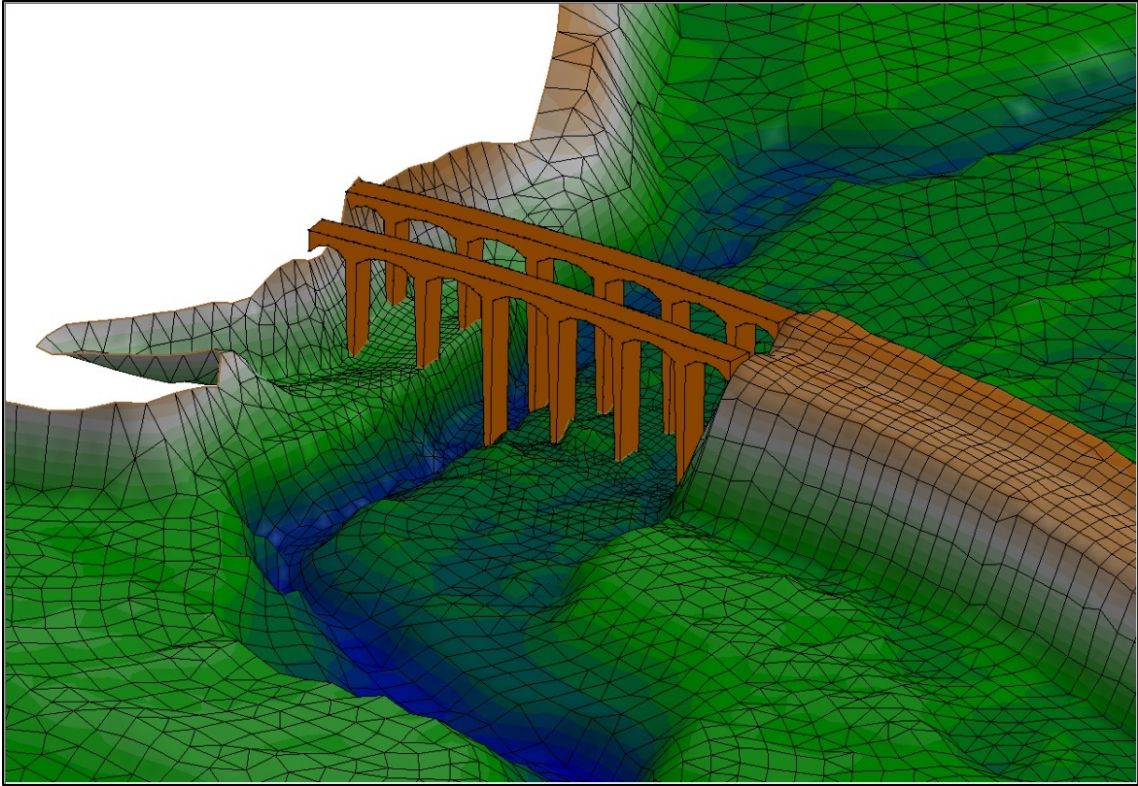
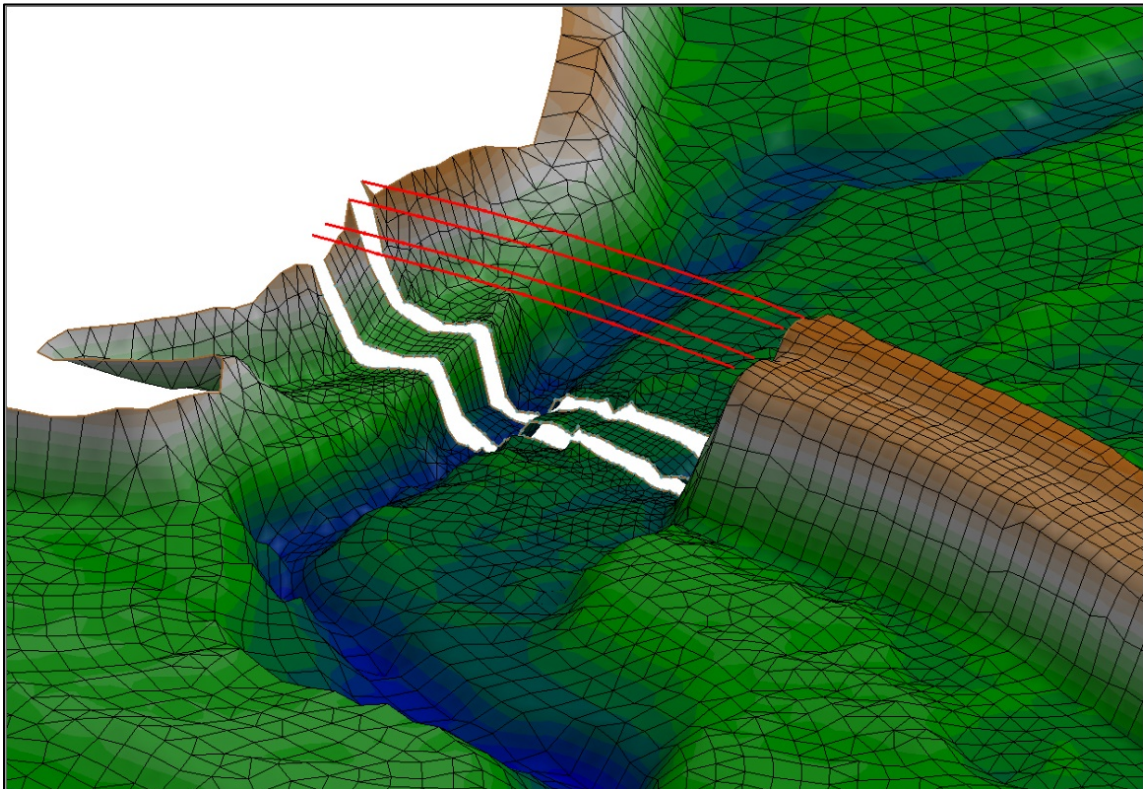Figure 5.22. Oblique view of mesh geometry and a bridge deck.

Figure 5.23. Oblique view of mesh geometry with mesh at bridge removed.

Another option in some models is to represent the bridge deck as an additional obstruction in the elements in the region of the bridge. This approach can be effective in generating backwater attributed to a bridge, but generally shoudl be calibrated to field data and provides no additional information about flow over verses flow through the bridge opening. The approach results in reduced velocities at the location of the obstruction.

A bridge is usually included in a model for one of two reasons. The first is to simulate the detailed flow and velocity distribution near the bridge and to understand the flow patterns. This is the case for design of a new bridge or analysis of an existing bridge, both for hydraulic design and scour evaluation. In these situations, the physical properties of the bridge are represented in the mesh. Not all models support all these features. For most transportation-related hydraulic analyses, it is important to represent the geometry of the bridge components and their influence on the flow as close to reality as possible. Selection of a model that has the appropriate features and capabilities for your project is very important. It is also equally important for the user to understand how bridge components are correctly represented in the selected model.

The second purpose of including a bridge in a simulation is to assess the upstream impacts caused by the backwater from the bridge constriction, and not the detailed flow characteristics through the bridge. In this case, the bridge is not the principal feature of the simulation, but its influence should be represented. Some models allow a simplistic representation of bridges for this case that represents the head loss through the structure but may not generate the detailed flow patterns through the bridge openings.

5.5.2. Culverts

Culverts may also be represented in a 2D model using any of the three methods described above. Culverts may (1) be the primary focus of a 2D model, (2) be a hydraulic feature included in a model, or (3) be so insignificant they do not warrant inclusion in a model. Representing culverts as 1D model features within a 2D model has been common practice. In this approach, the 2D model computes WSEs at the upstream and downstream end of a culvert. These water levels are combined with the definition of the culvert (number of barrels, barrel size, and shape, entrance type, material type, etc.) to compute the conveyance through the culvert. This result becomes a local boundary condition in the 2D model with flow leaving the 2D domain at the upstream end of the culvert and re-entering at the downstream end of the culvert. Because the culvert may impact the upstream and downstream WSE, this approach is typically iterative. This approach is, in effect, a 1D or analytical model of the culvert that communicates with, or is integrated into, the 2D model. This approach is limited by the method used to compute flow through a culvert by the model. It may ignore the velocity head of water entering the culvert since most culvert analysis assumes still pool conditions, and it does not represent the variation of the flow inside the culvert itself.

Another option for representing culverts is to represent them similar to a bridge, as described in the section above. In this case, the user incorporates the lower half, or floor of the culvert geometry into the mesh and represents the top of the culvert as a bridge deck (see Figure 5.24). The vertical sides of the culvert can be represented as very steep banks in the mesh, or as holes in the mesh, depending on the model used. This approach involves more work in developing the mesh but has the advantage of representing the 2D flow field through the culvert and accounting for upstream velocity. This approach is currently recommended for box culverts and arch culverts, as it is much more challenging to represent the geometry of cylindrical shaped conduit in a mesh.
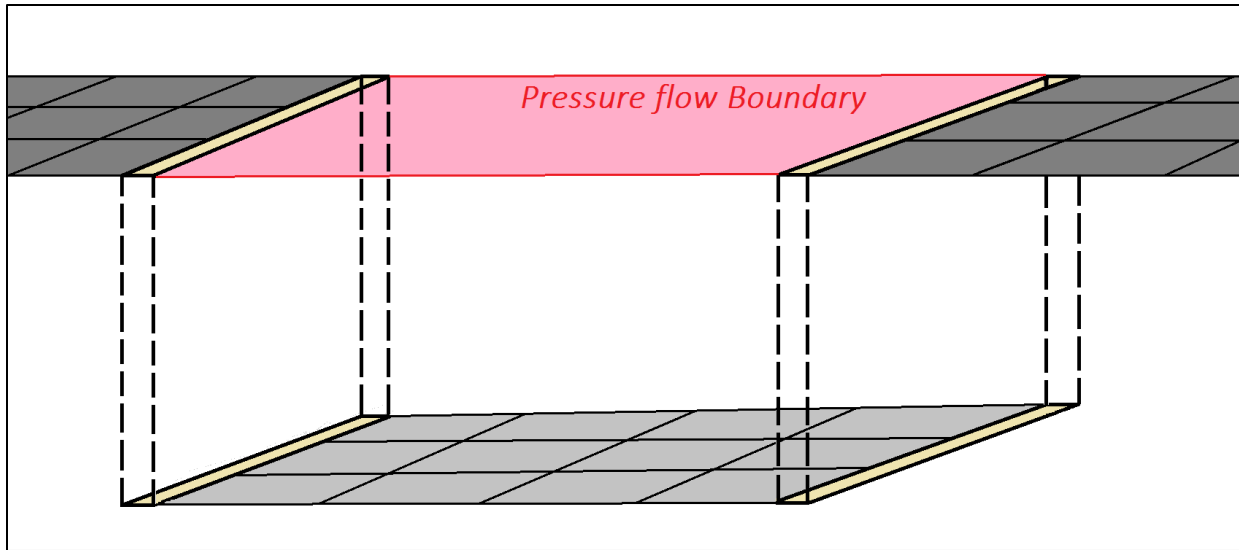
Figure 5.24. Culvert represented as a pressure zone with vertical walls.

5.5.3. 1D Features

Additional types of hydraulic structures can be included in a 2D model similar to the 1D culvert approach. If flow rates through a structure can be represented with an equation based on some local geometric parameters such as a crest elevation or an orifice diameter and some generalized coefficients such as a weir coefficient or inlet type, their impact on flow in the domain can be simulated as a 1D calculation. This technique applies to sharp crested or complex weirs, gated structures, links, spillways, orifices, and other hydraulic structures. These methods can be used to simulate storm drain networks. For some models, the storm drain network can be incorporated as a separate 1D model connected to the 2D model at each entrance, manhole, and outfall. Various levels of this capability are available in many, but not all 2D models. The introduction of these types of structures can introduce instabilities and may necessitate additional attention to mesh quality to achieve a stable solution.

5.5.4. Buildings

When buildings and other structures that cause flow obstructions are present within a model domain, the modeler has choices for how to represent such features. Four approaches are typically employed, depending on the model used, including:

- Represent the urban/developed region as a composite area with an increased Manning's roughness value
- Specify an attribute for elements inside of the building as obstructions that compute additional drag and reduces flow through those elements (blocked obstructions)
- Represent individual buildings with a high Manning's roughness value and ensure that mesh elements near the feature are similar in size and footprint.
- Specify holes in the mesh at individual building locations

The first approach requires less effort and follows a standard modeling methodology. As shown in Figure 5.25, the cross-hatched (Developed) region represents a developed region with buildings and other flow obstructions such as fences, vehicles, sheds, and trees. By specifying a larger Manning's n value for this region, the model can account for the impact of reduced flow through the region. However, this approach does not capture flow variations along roads and

around buildings, and without calibration data, assigning a Manning's roughness value to represent the composite roughness may be difficult to determine and verify.
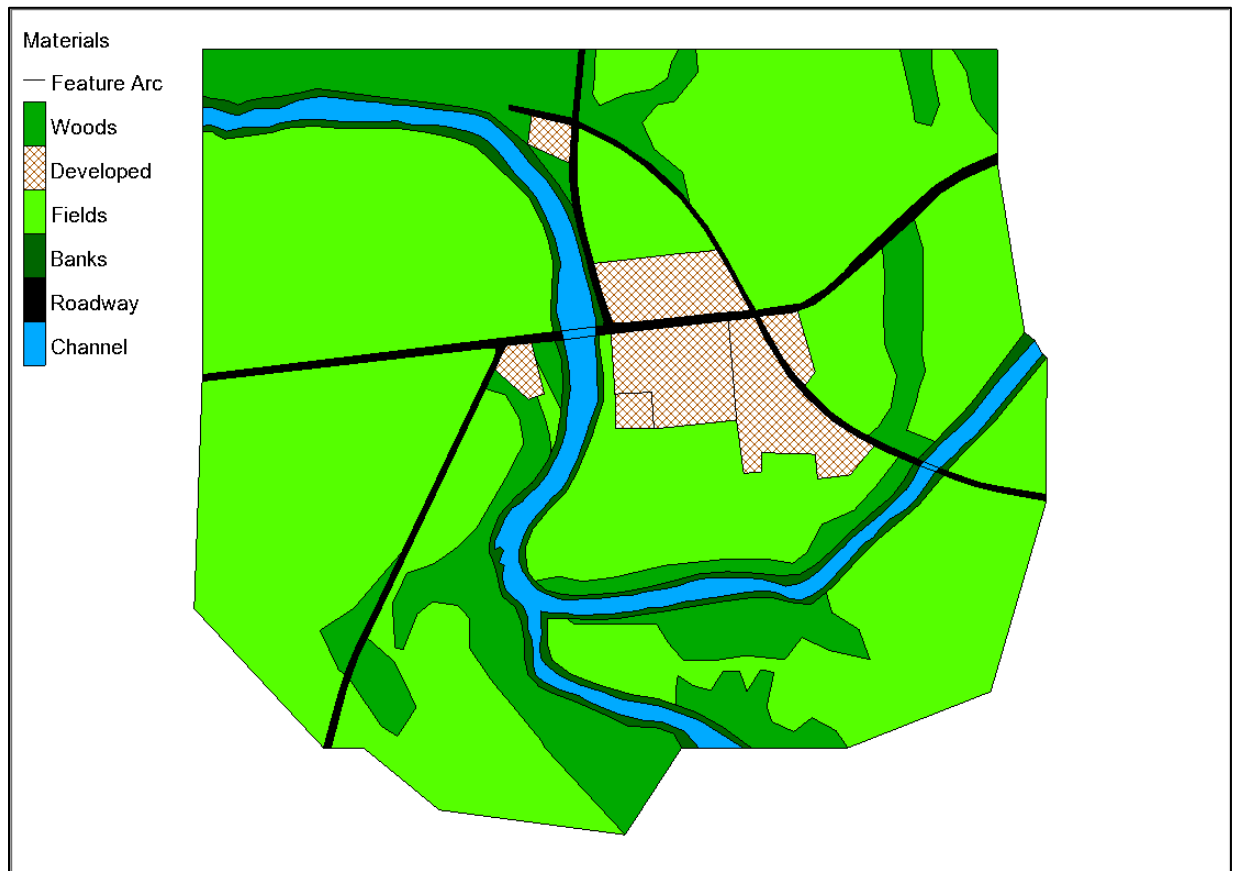
Figure 5.25. Spatial representation of material zones.

The second approach provides the modeler with a way to incorporate detail without having to create as dense a mesh as is required with the third and fourth approaches (described below). In the second approach, individual buildings can be represented as obstructions (geometric points, lines, or polygons) with attributes such as porosity and height. The model then computes additional drag associated with this obstruction and reduces the conveyance through the elements containing the obstruction. This mimics the impact of conveyance caused by the building but does not simulate the flow patterns around the building. This capability is currently limited to a few models.

The third and fourth approaches require the building footprints to be incorporated into the mesh. In the third approach, individual buildings can be represented in the land use map. Just as in the first two approaches, the spatial representation of the land use types used in the model are assigned to individual elements. However, in this approach, the elements correspond to a single building rather than a conglomerate of multiple buildings and possibly space in between. (Creating a highly detailed land use data set is not necessary unless the associated mesh elements include a similar level of detail.) The Manning's n value for the building type can be set high enough to represent limited or no flow through the elements representing the buildings. The fourth approach represents building footprints as voids in the mesh and provides the most detailed representation of the hydraulics in locations with buildings (Figure 5.26).
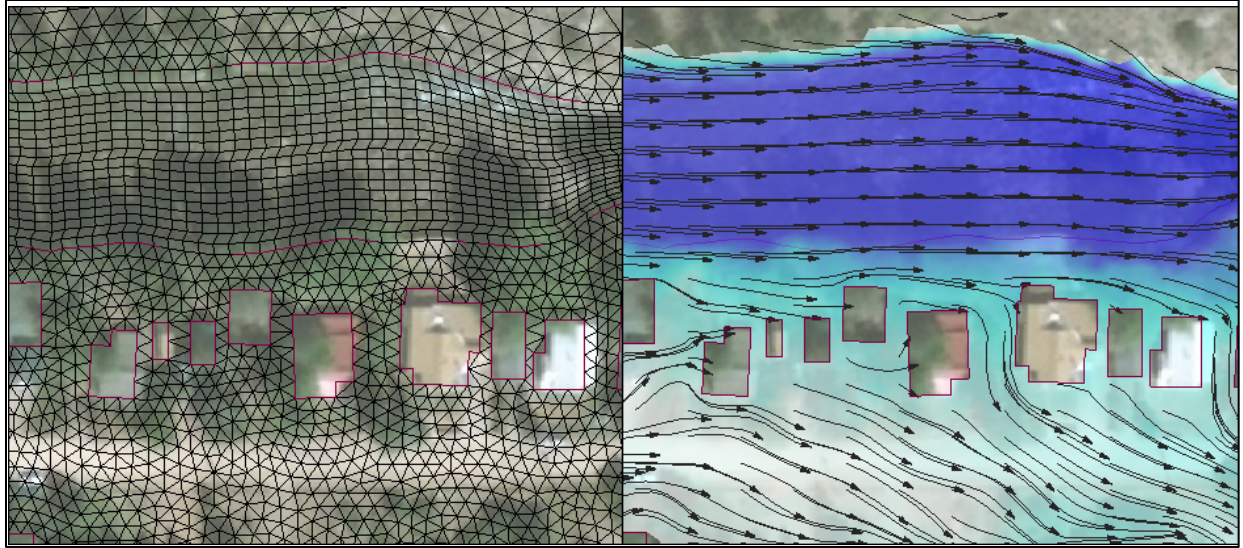
Figure 5.26. Representing buildings as voids in the mesh.

In the first two methods, flow is still allowed to move through the elements containing the buildings, but the higher flow resistance provides an approximated reduction in conveyance on the hydraulics. These methods are acceptable if detailed hydraulics are not needed near the buildings. The third and fourth approaches, allow for representation of flow patterns around the buildings. The increased level of detail and model resolution required to represent buildings within the mesh often deters modelers from using this approach unless necessary.

5.6.   LAND USE TYPES (ROUGHNESS)

The effect of vegetation, or other surface features, on flow, comprises one of the most significant factors involved in a model. This factor is generally referred to as roughness or bed friction and should reflect physical conditions. A sandy channel bottom provides much lower resistance to flow than a brushy field, which in turn provides less resistance than dense crops or heavily wooded areas. Numerical models typically allow specification of resistance that varies as required to match physical conditions through the domain. The most common approach allows the specification of roughness for every computational point or element in the domain.

To simplify the procedure and to assist in providing consistency, it is common to develop a list of land use types and associated roughness parameters that exist in the domain (i.e., channel, field, forest, urban area, etc.), or obtain GIS data delineating different land use types. This approach allows the modeler to specify the numeric parameters (Manning's n, Chezy C, friction coefficient, etc.) for each land use classification or "material type." The modeler can then map a material type to each element in the domain. This approach requires that the modeler applies a consistent specification for areas of similar attributes.

Areas representing the land use types can be delineated using an appropriate orthoimage of the project area. Similar to the mesh geometry, the resolution of land use type delineation can vary from a general representation to a very detailed delineation, as shown in Figure 5.27. The modeler should decide how much detail is required at locations throughout the model domain based on the model application. For a bridge analysis project, the land use types delineation should be more detailed near the bridge and roadway and can be less detailed as the model domain extends upstream and downstream of this location.
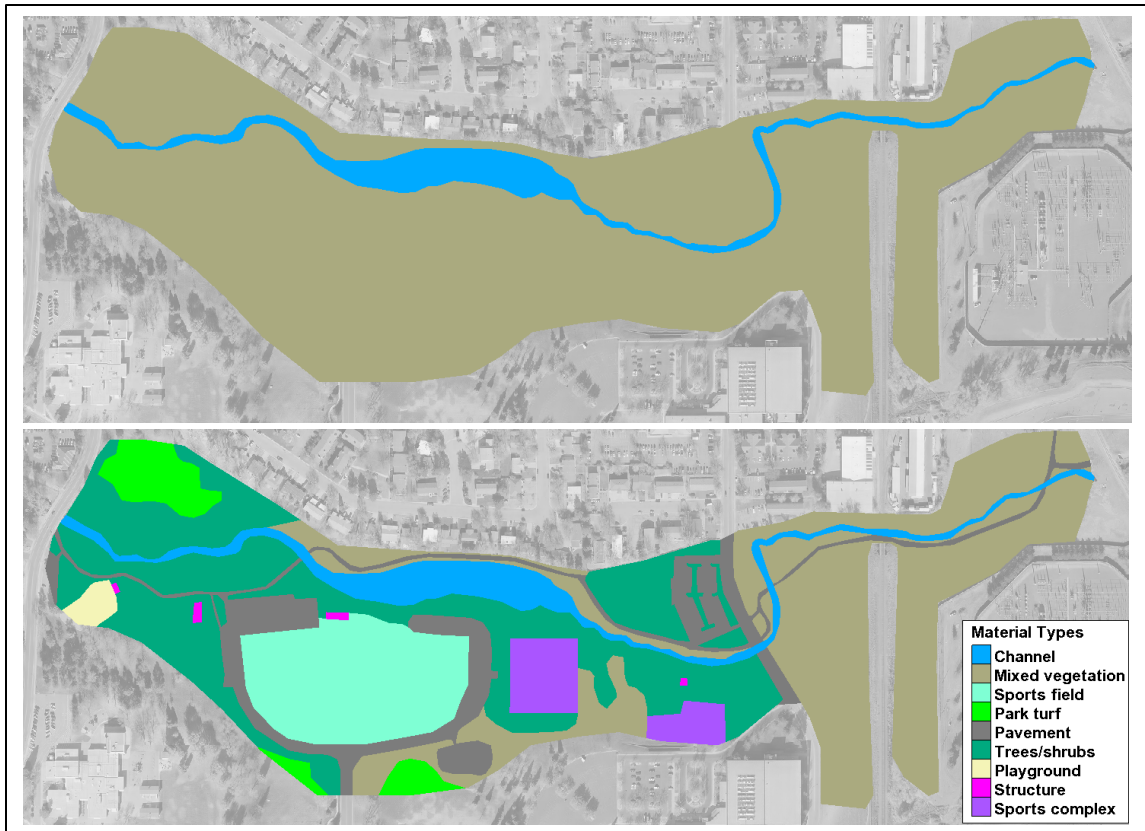
Figure 5.27. Land use type delineation examples; general (top) and detailed (bottom).

5.7. MONITORING POINTS AND LINES

Numeric models and software applications often provide the ability to extract hydraulic data from the domain at specified locations. These locations consist of lines that span sections of the domain (referred to variously as flux lines, continuity check lines, monitor lines, etc.) and specific point locations in the domain (referred to as monitor points, gauges, etc.). By outputting computations at these locations, a model provides useful information to verify performance, stability, and results. Typical uses of monitor points and lines include:

- Evaluating convergence of a water level to a stable value (calibration/verification)
- Evaluating conservation of mass around a portion of the domain such as a bridge structure
- Computing percentage of flow in various channels and flow paths

During the model development process, the modeler should determine what hydraulic variables are required for the analysis, where those values should be monitored within the model, and what method should be used to obtain them. See Chapter 6.1 for further discussion related to using monitor points and lines.

5.8. MODEL CONTROL AND EXECUTION

In addition to the geometric definition, boundary condition specification, and material attribute assignments, most numeric models require other parameters to control model execution. The following sections cover standard model parameters modelers may need to consider before model execution.

5.8.1. Initial Conditions

The initial model condition is a user-specified starting point required for numerical computations to begin. The initial conditions required for 2D models include the WSE and u and v velocity values at each element.

The initial condition can be simplified to one of two cases: a "flooded" (bathtub) or "dry" mesh. Either of these conditions specified by a user are often referred to as a "cold-start." Alternately, initial conditions can consist of a complete definition of a hydraulic flow field (velocities and water depths throughout the mesh). This condition is often referred to as a "hot start." Specifying initial conditions close to the final solution often improves the stability and performance of a numeric model.

Many early models required initial runs to start with "bathtub"-type conditions in which the domain is completely inundated with zero velocities. This inundated, stagnant condition is numerically stable and allows the model to apply the forcing boundary conditions incrementally to create realistic flow conditions.

As numeric methods have improved, some models now offer the ability to assign a dry initial condition. This approach starts with a dry mesh and allows the water to flood the domain from inflow to exit. If the domain includes significant floodplain storage, this method may need significant computation time to reach steady-state conditions.

Manually defining a complete hydraulic flow field is not practical. Therefore, most numeric models now support the use of a hot start initial condition. A hot start uses previously modeled solutions as a starting point instead of dry or inundated conditions. Hot starts are advantageous for providing initial conditions for model simulations where a significant amount of computing time is anticipated. A hot start can help reduce model instabilities by allowing enough "warm-up" time at the start of a simulation. When modeling floodplains with large amounts of storage potential such as backwater areas, topographic depressions, lakes, and other storage, it may be more efficient if initial water surfaces are present before the introduction of peak flows.

A user may want to run a single simulation and use it to provide initial conditions for successive model simulations. Hot start files are helpful if an analysis requires small changes to model parameters during subsequent model simulations. Changes may include adjustments to node elevations or material coverages. Additionally, it is possible to reduce computational time by running an initial simulation using an initially larger time step; then subsequent simulations can use a hot start file using with a smaller time step to achieve model stability.

5.8.2. Timing

Depending on the model used and the type of analysis, the modeler may be required to specify timing parameters. Most often, these include run duration and a computation time step. Appropriate time steps vary based on the model and the dependence on CFL type conditions (see Section 3.5.2).

Model duration depends on the size of the domain and the amount of discharge to be analyzed. For steady-state applications, the model should be run long enough to reach convergence. This requires enough simulation time for the flow to enter, pass through and exit the domain as well as fill all storage areas. When determining convergence of this type, modelers can compare flow entering and leaving the domain to make sure they match. They can also compare the last two

output flow fields for a steady-state simulation. If the mass of water in the simulation for those two states match, the model has reached steady-state conditions.

When the simulation includes dynamic conditions, meaning the simulation of a hydrograph or tidal cycle, the duration of the simulation should include enough time to reach preliminary conditions and then the entire event being modeled. The preliminary conditions are often reached in a preliminary run, and a hot start file is saved. The simulation duration for a hydrograph should include the duration of the flood itself plus time for the domain to drain. For tidal simulations, it is best practice to include multiple tidal cycles with the first one or two being used to establish a transient convergence. Common tidal applications could require a full lunar tide (28 days) of simulation.

5.8.3. Physical Properties

Most numeric models require the specification of physical constants, including terms such as the acceleration due to gravity and water density. Some of these parameters, such as kinematic eddy viscosity, are often numeric parameters rather than physical ones. Kinematic eddy viscosity does not equal the physical property of viscosity.

In addition, each model defines a method or methods for configuring the partial differential equation terms in the conservation of momentum equations (Section 3.2). To simulate the turbulence that forms in flow, first-generation models (RMA2 and FESWMS) typically required the specification of spatially varied kinematic eddy viscosity or Peclet number. Common practice included using large eddy viscosity values and incrementally lowering them as far as numeric stability allowed.

Latest generation models have improved to the point that this is not needed and is sometimes not an option. Instead, latest-generation models can provide a turbulence coefficient with a recommended value that should not be changed unless significant calibration data is available such as in a flume. The simulation of turbulence is the most complicated portion of the numerics involved in these models. Care should be taken to follow recommendations from the developer of the model being used to understand what parameters mean and how they should vary.

5.8.4. Model Execution

Execution requirements vary from model to model. Most models require some preliminary computations to format data or prepare supplementary information from the geometry or other model parameters. Details of how a specific model operates should be available in the documentation for that product. The user generally should specify model parameters to define the type and frequency of data output.

Most models require the items covered in this chapter, including:

- Mesh geometry representing the terrain (Section 5.3)
- Boundary conditions (Section 5.4)
- Hydraulic structure features important to project (Section 5.5)
- Land use type coverage representing roughness types (Section 5.6)
- Monitoring lines and points (Section 5.7)
- Model parameters (Section 5.8)

CHAPTER 6

MODEL REVIEW AND CALIBRATION

Hydraulic models need thorough reviews and calibration and verification to known historical data when available, before using them for analysis. For this reference document, definition of these terms are as follows:

- Calibration – The process of adjusting model variables, constants, and parameters until the model output represents some observations (recommended for project application if data is available).

- Verification – The process of determining how well a calibrated model agrees with some observations (optional step if additional data is available).

Model reviews should occur multiple times throughout the model development process, not only once after the model has been developed, run, and results generated. Evaluation of model results is a crucial part of the review process but should not be the only review of a 2D model. Assessment of the appropriate model parameters during the development of the mesh helps reduce the number of potential errors in the model results. Once a model has been developed and run, the modeler and the reviewer should understand how to view and evaluate model parameters and results. A good understanding of how to view and evaluate results will aid in determining any necessary model modifications.

After an initial model review and evaluation of results has been completed, model parameters should be calibrated to known hydraulic data if available. Calibration data allows the modeler to check the model results or adjust model parameters to match measured data within a desired tolerance. A poor model can be adjusted (sometimes erroneously) to calibrate to known data, but that does not mean the model is producing useful results.

In the absence of calibration data, and even under specific scenarios where there may not be sufficient calibration data, a sensitivity analysis of model parameters should be conducted. Sensitivity analysis determines the relative effect of model parameter changes on simulation results. The sensitivity analysis provides the modeler with more confidence in the final results.

6.1. EVALUATING RESULTS

After completing a simulation, model results should be reviewed to determine if the simulation was successful. Before a simulation is considered final and the results are used for analysis, design calculations, or provided to others, the modeler should perform a standard and repeatable review process. This review should include the following items that are expanded on in the next sections:

- Stability of solution
- Continuity of flow through the model
- Reasonableness of flow regime (Froude Number)
- Representation of hydraulic structures
- Conservation of mass or mass balance

6.1.1. Stability

It is important to verify that model results are stable no matter what type of conditions are being analyzed. A model solution is stable when it does not exhibit significant changes from one time step to another. Stable solutions are one indication a 2D model has run successfully. Stability for older 2D models was a significant issue and often required considerable effort to arrive at a stable solution. Newer generation 2D models are significantly more stable than previous models. Arriving at a stable solution is no longer a lengthy, iterative process, but model stability still needs to be checked.

Model stability can be evaluated by qualitative and quantitative methods. Qualitative evaluations are conducted by viewing model results between consecutive time steps and determining if results appear stable (Figure 6.1). In this figure, the velocity results from a steady-state simulation, are oscillating at multiple locations as seen by viewing consecutive time steps. Qualitative assessments are sufficient for preliminary model runs. However, quantitative evaluations should be made for final model results.
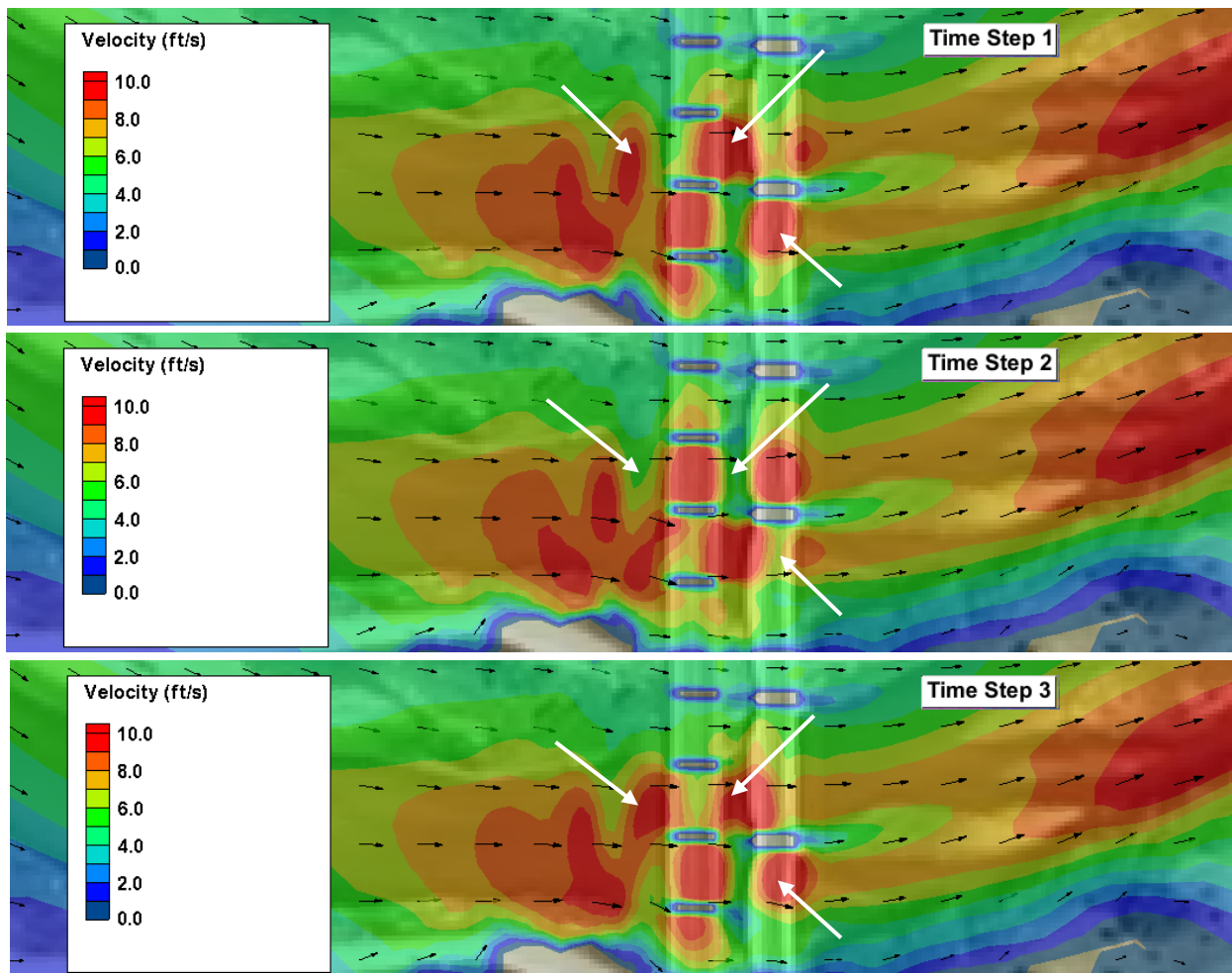
Figure 6.1. Example of qualitative instability comparison plots showing changing velocity over different model result time steps.

A quantitative evaluation of stability may include assessing changes in hydraulic variables at monitor lines, points, elements, nodes, or other specific locations within the model geometry where instability is a concern (see Figure 6.2). In this table, the discharge and WSE across a monitor line at the upstream side of the bridge from a steady-state simulation are changing. Both variables should reach equilibrium in a stable, steady-state model.

Figure 6.2. Example of quantitative instability assessment (output from bridge pressure flow lines at upstream side of bridge, note unstable WSE and Q).

For unsteady flow simulations where the WSE and discharge are expected to change it is good practice to run a steady-state simulation across the range of the unsteady hydrograph to determine if your model is able to achieve stable flows. Also, some models have the ability to check mass conservation for each time step of a model. This allows the modeler to compare if the total mass entering the model domain is equal to the total mass in the domain, which helps assess model stability for an unsteady run.

Common reasons for 2D model instability include:

- Attempting to model hydraulic situations outside the limitations of the model such as locations that would typically exhibit highly turbulent flow or extreme slopes
- Introducing additional computation requirements, such as bridges, culverts, weirs, etc.
- Time step is too large
- Mesh quality issues
- Model was not run for sufficient time
- Terrain issues

If model results are not stable, the modeler should determine the cause of instability and make appropriate adjustments to the model settings, geometry, or other parameters. Common solutions to the sources of instability listed above include:

- Assess need for 3D model
- Reduce time step up to one order of magnitude
- Increase run times until model stability is achieved
- Adjust geometry or model settings until a stable solution is achieved
- Review/adjust hydraulic structure input data, adjust geometry data
- Address terrain or mesh quality issues
- Increase element size
- Understand model limitations

6.1.2. Continuity

When modeling a steady-state condition, the discharge measured throughout the entire model should generally be within about 2% of the inflows designated at the upstream boundary. Complex models with multiple internal boundary conditions may be an exception. The modeler should check for continuity at multiple locations throughout the model. One method for checking continuity is by using monitor lines that calculate flow across a designated line for each timestep (Figure 6.3). It is good practice to check continuity at multiple locations throughout the model domain.

If the model includes split flow, it is important to assess the continuity through each flow path. If a model simulation includes a dynamic hydrograph or is a tidal model, the discharge is not likely to be constant throughout the domain. For unsteady modeling, it is important to verify the mass conservation of the model as discussed above for model stability. Discharges throughout the domain should be reviewed for reasonableness, even though a direct correlation to the inflow discharge is not expected.
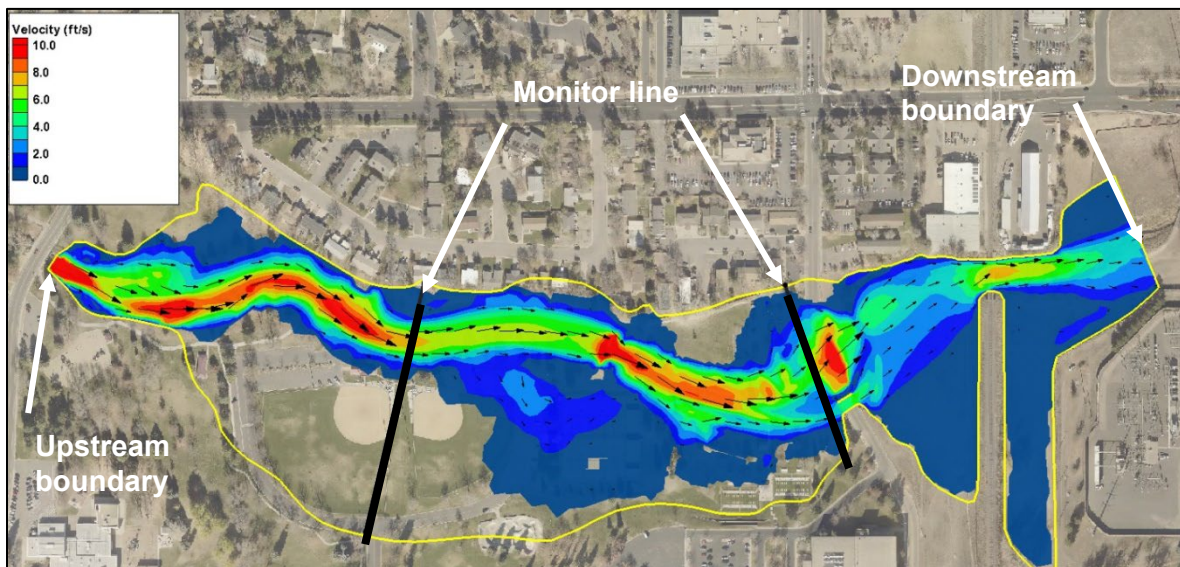
Figure 6.3. Monitor lines used to assess continuity.

The modeler should determine locations within the model to check continuity for each simulation run, in addition to the discharge reported at the downstream boundary. Software applications can be used to extract hydraulic variables such as discharge at specified locations within the 2D domain. If continuity is not achieved, adjustments to the model control settings and the underlying mesh should be made until discharge throughout the model is within acceptable an acceptable percentage of the inflow(s).

6.1.3. Froude Number

The Froude number (Fr) is helpful in reviewing the reasonableness of model results. The Fr is a dimensionless number that describes the flow regime of open channel flow (see Table 6.1 and Figure 6.4). The equation to determine Fr is:

$$Fr = \frac{V}{\sqrt{gH}}$$

(6.1)

where:

V = velocity of flow
H = depth of water
g = gravitational acceleration

Table 6.1. Flow regime related to Froude number.

Froude Number	Flow Regime
Fr < 1	Flow is subcritical
Fr = 1	Flow is critical
Fr > 1	Flow is supercritical
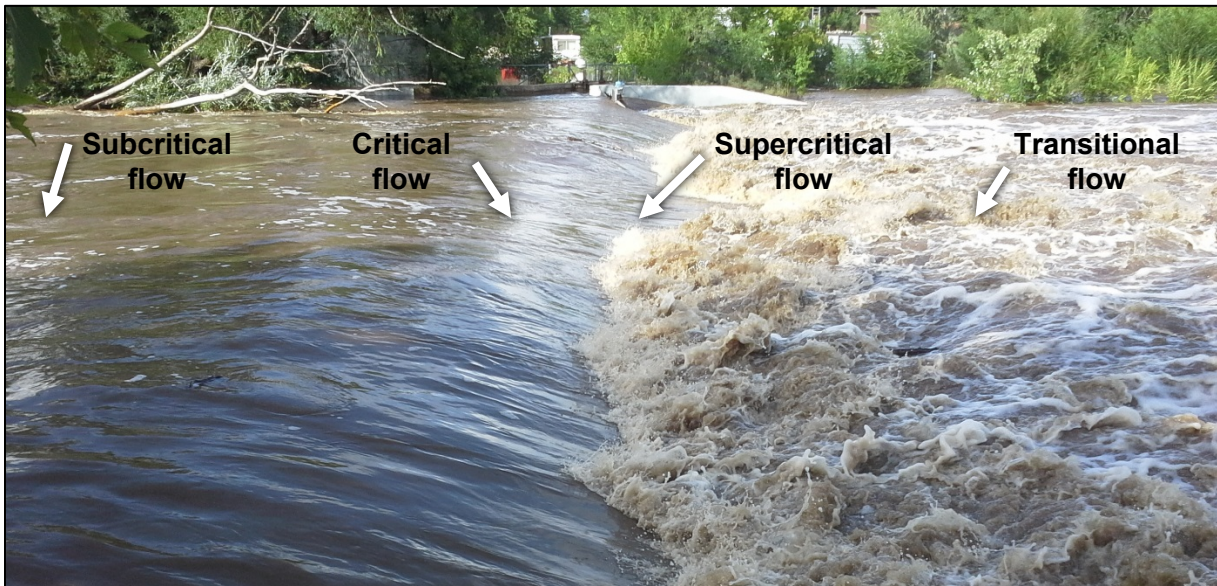
Figure 6.4. Examples of subcritical, supercritical, and critical flow.

The Froude number is a result calculated by some 2D models (see Figure 6.5). If the Froude number is not provided by the model, it can be calculated manually from the velocity and depth results. In natural river systems, subcritical flow is the dominant flow regime, and critical and supercritical flow do not typically occur for extended lengths.

Supercritical flow is not normally sustainable for long reaches in a natural system because the transition from high potential energy associated with subcritical flow to the high kinetic energy associated with supercritical flow typically changes the channel (boundary) characteristics. Two

primary drivers control the flow regime. First, as velocities increase, and bottom material and channel cross section vary, the flow is typically highly turbulent flow, which results in increased resistance. Second, as flow transitions from sub- to supercritical, the depth decreases significantly, thereby increasing the velocity. As this occurs, natural channels degrade vertically and laterally until the flow regime moves back to subcritical flow, thereby reducing the kinetic energy of the water.

Locations where modelers may expect critical and supercritical flow include:

- Flow overtopping roadways
- Weirs and low head dams
- Isolated locations within steep river systems

"If the supercritical flow is indicated in the hydraulic analysis for long lengths of channel, a re-evaluation of roughness coefficients probably will show all energy losses were not accounted for (Jarrett 1985)." If model results calculate supercritical flow over long sections of the main channel or floodplain, model parameters, typically Manning's roughness values, need to be adjusted until this does not occur. Smaller time steps may be necessary if attempting to model supercritical flow and hydraulic jumps.
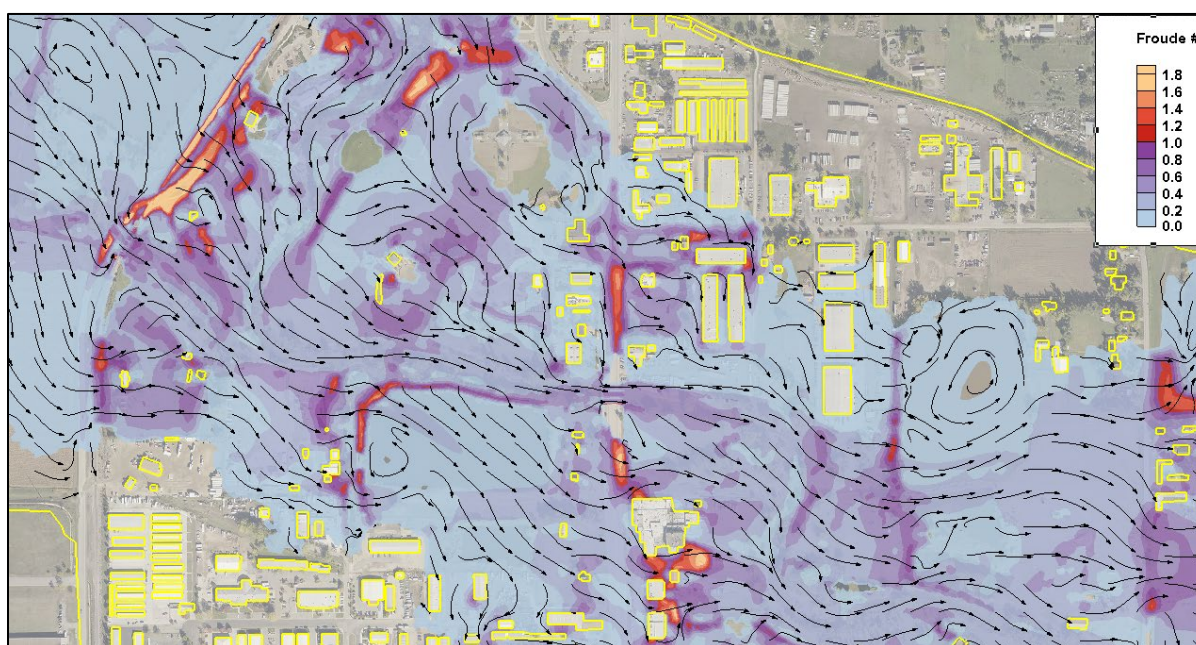
Figure 6.5. 2D model results of Froude number.

A situation where adjustments to the hydraulic model may be required, to avoid predominately supercritical flow in the channel, exist in steep mountainous river systems. The standard approach of applying roughness coefficients often results in long reaches of supercritical flow in high gradient streams.

In this situation, an approach developed by Robert Jarrett with the USGS can be used to artificially increase the roughness value to account for the increased turbulence in steep streams. More information on this approach can be found in "Determination of Roughness Coefficients for Streams in Colorado" (Jarrett, 1985).

6.1.4. Structures

If a 2D model includes hydraulic structures such as bridges, culverts, gates, or weirs, modelers should assess the results at each structure. It is the responsibility of the modeler to determine if the structure is represented correctly in the model geometry, that structure parameters were entered correctly in model settings, and that the model results are reasonable.

Introducing hydraulic structures into a 2D model adds computational complexity and can be a source of discontinuity, instability, or incorrect representation of hydraulic results. Because structures can be a primary hydraulic control or the focus of hydraulic analysis, it is important to evaluate the model results near each structure.

Hydraulic results at structures can be evaluated by plan view, cross section, and profile results as well as tabular data provided in model output files associated with each structure and at additional monitor points or lines set up by the modeler (Figure 6.6).
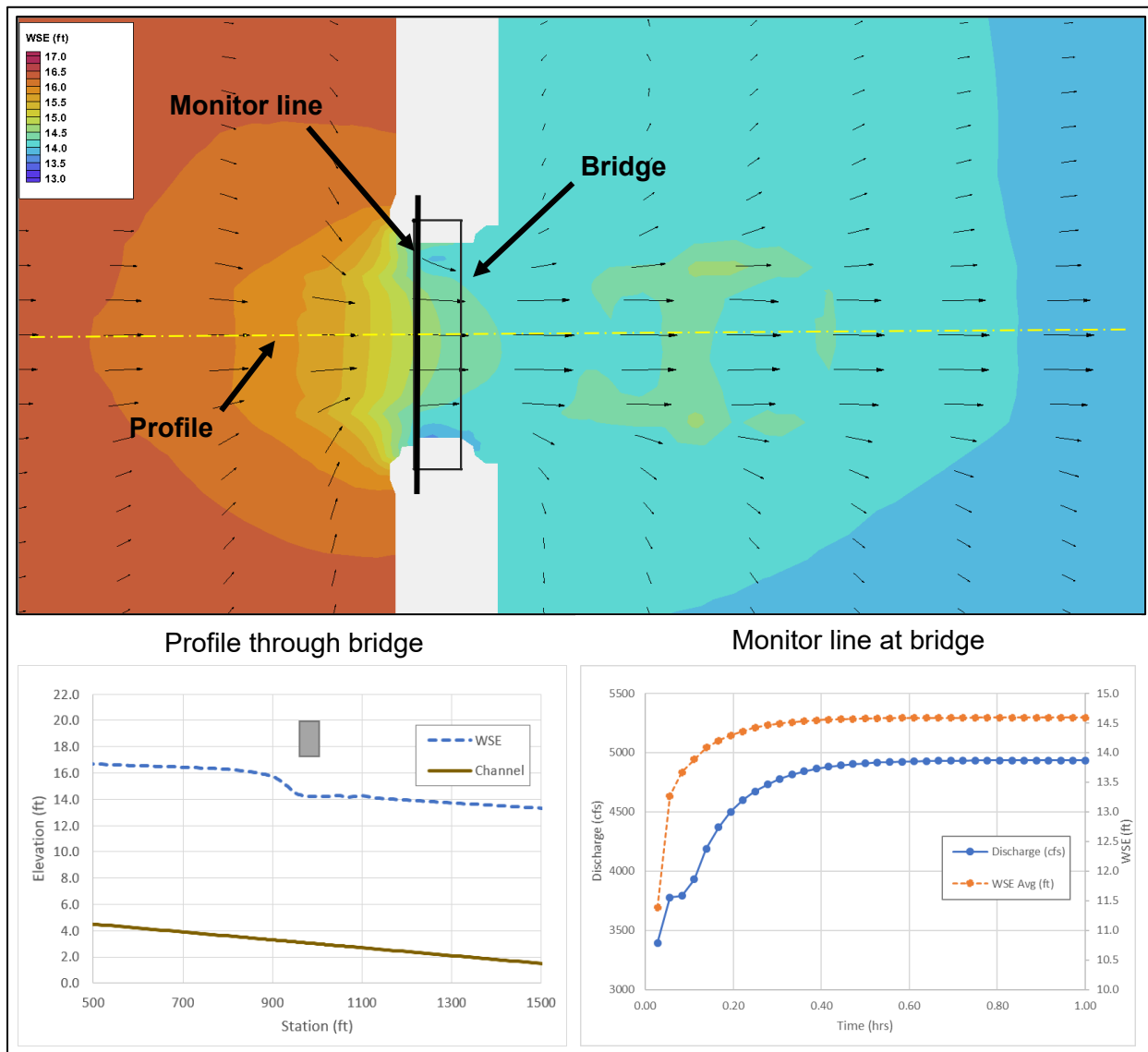
Figure 6.6. Plan view and profile of WSE and monitor line review for bridge opening.

6.7

If results do not appear reasonable, offline calculations should be conducted as a secondary check. Typical hydraulic variables to be evaluated include:

- Impact to the water surface, including head loss, through a structure
- Discharge through structure
- Velocity in the vicinity of a structure

6.2. THE CALIBRATION PROCESS

A typical model calibration process starts with the collection of observed data from a historical event near the project area. Upon completion of a model simulation, comparisons can be made between computed WSEs and observed values. If the computed WSE results generally match the observed data, then the model is calibrated. If there is a difference greater than an acceptable amount, then the model parameters should be adjusted, model geometry reviewed and adjusted as required, and the model simulation rerun. This process is repeated until the comparison of the observed values and the computed values are within an acceptable level. An acceptable level of difference needs to be determined by the modeler while considering the application of the model results, and reliability of the calibration data.

It is common for the terrain data used to create the model geometry not to match the terrain data that existed during the historical event for which calibration data is available. If additional observed data from a separate event, or a different data type from the same event, is available, the calibrated model should be verified with the additional data.

When calibration data is limited or not available, sensitivity testing should be performed on models with parameters of assumed or inherent uncertainties. Sensitivity testing is a process of adjusting model parameters or characteristics that have inherent uncertainty and observing changes in the model results in the area of interest. As is the case with model calibration, testing the sensitivity of model parameters requires the modeler to determine the appropriate value of those parameters to provide reasonably conservative results. This process is not only a review of results after a model run but could also be an iterative process during model development (see Section 6.4 for more information).

6.3. DATA TYPES

Calibration data may be available in many forms, such as measured data, firsthand accounts, historical records, and field observations. It is up to the modeler to best interpret the available calibration data for use in the hydraulic model. Typical calibration data is a field measured value of WSE, depth, or velocity during or following a specific flow event.

The USGS has several resources to assist in the identification and understanding of high-water mark data, as well as online resources that provide access to data from historical events. Two helpful resources include;

- Identifying and Preserving High-Water Mark Data (USGS 2016)
- USGS Flood Event Viewer

Some types of calibration data are more useful than others and have the potential to provide better information for model calibration. The reliability of any calibration data depends on the source of the data and collection methods. Typical sources include:

6.3.1. Water Surface During Calibration Event

Gauge data. The USGS maintains a nationwide network of stream and tidal gauges that record water stage. The modeler can translate the recorded stage information to a WSE if the datum elevation of the gauge is known. The WSE is the measured value, and the discharge is calculated from the WSE measurement. The reliability of the discharge value is dependent on the accuracy of the rating curve used to determine discharge, and how often the gauge is calibrated. Gauges can get damaged, destroyed, or impacted by debris in large flood events, so modelers should understand the potential limitations of the recorded data during high flow events. Other federal, state, and local agencies also collect gauge data during significant weather events and may have additional data for assisting with calibration.

Mud or debris lines. After a flood event, mud lines on buildings and other structures can provide a peak WSE at a known location (Figure 6.7). Similarly, flood events often deposit debris that can help determine peak WSE (Figure 6.8). The high-water elevation can be obtained through survey data or measured from the adjacent grade to the mud line utilizing the elevations from a quality terrain surface for the adjacent grade.

High-water mark photos. Georeferenced photos of hard surface locations like a sidewalk or road can be used in conjunction with aerial photography and a DEM to determine the elevation of the point in the photo (see Figure 6.9). The key for using this method is a high accuracy terrain surface and high-resolution orthoimage to locate the point in the photo.

First-person accounts. Another possible source of high-water information can be first-person accounts. First-person accounts can be obtained by talking to landowners or other observers. Verifiable photos or video from a third party are also useful if the location, date and time the photo or video is known. First-person accounts are more susceptible to errors and exaggeration the further the event was in the past. Collecting these accounts as soon as possible after an event is important to minimize any error in the account.

Figure 6.7. Mud line on building indicating high-water line.

Figure 6.8. Flood debris on a bridge deck.

Original Photos: © 2019 Google® (see Acknowledgements section)

Figure 6.9. Site photo of high-water line and associated orthoimage.

Nails and tape. Nails and tape, representing a WSE at a specific time, can be driven into or attached to a tree or telephone pole during an event, or tape can be tied around a tree, post, or other fixed objects. As with the first-person accounts, it is critical to know the date and time the nail was driven so that the mark can be correctly associated with an event. If the timing of nail placement can be verified and correlated with a discharge, these marks provide a reference for a surveyor to obtain elevation data.

Use of other models. It is possible to calibrate a new model against older model results. This comparison can be to a 1D model or a 2D model. The modeler needs to be very confident in the accuracy of the comparison hydraulic model.

6.3.2. Discharge Related to Calibration Event

All calibration data represents a hydraulic variable, typically WSE, of a specific event. For calibration data to be useful, the discharge associated with the data needs to be determined. Gauge stations have been calibrated to convert the measured stage to a flow rate. If a gauge is within the project area, the gauge can be used to help determine the discharge of a historical event. However, modelers should remember that gauges measure the water level and calculate a discharge from the measured value.

The process of determining the associated discharge is not straightforward and should consider the following dynamic characteristics of natural river systems:

- Flows are typically unsteady.

- The terrain of the project area may be difficult/impossible to replicate due to a calibration event occurring in the past and associated erosion or sediment deposition that can occur in a channel and floodplain during high flow events.

- High flow events typically have debris associated with them and estimating the hydraulic impacts of debris can be difficult.

Approximate calibration data and the associated discharge are better than having no data at all and can still provide some level of calibration.

6.4. SENSITIVITY TESTING

Sensitivity testing of various aspects of hydraulic models may be necessary to check model results. The sensitivity testing process includes the following steps:

1. Adjust estimated model parameters (up and down).
2. Observe changes to the model solution.
3. Compare changes to pre-adjusted model condition.
4. Determine preferred parameters for specific analysis.

Any model parameters not confirmed with calibration data may need to be tested to determine confidence in the model results at the project area. Initial adjustments to parameters should be made within the extreme ranges of reasonable error. This step helps the modeler better understand how the model reacts (i.e., is sensitive) to specific changes. Secondary fine-tuning should include smaller adjustments until the modeler has confidence in the results.

Parameters that have minimal effect on the model solution are considered insensitive, and parameters that result in changes to the model solution are considered sensitive. Modifying sensitive parameters should result in increased confidence in the model solution when calibrated against known data. For parameters that provide a varying range of solutions, modelers should use values that are within a reasonable range and provide the most conservative solution. A single value for a parameter may not provide the most conservative solution for all considerations. For example, using a roughness on the high end of that reasonable range may be conservative for estimating water surface elevations, but using a roughness on the low end of the range might be most conservative for estimating velocities. The following paragraphs provide a discussion of the sensitivity testing of model parameters that typically involve assumptions or inherent inaccuracies.

6.4.1. Upstream Boundary Sensitivity Testing

The upstream boundary location, width, and parameters include inherent inaccuracies. An upstream boundary condition may have a known discharge, but the method by which the discharge is distributed at the edge of the mesh may influence results downstream for some distance. If a sensitivity analysis reveals significant differences in results at the area of interest when the width or location of the upstream boundary condition is varied, the modeler should find an appropriate balance. A depth-velocity solution (as described in Section 7.4.1.) should be developed to help visualize the flow distribution and assist with determining appropriate upstream boundary settings. In addition, assessing how the model distributes flow helps evaluate the parameters that control flow distribution at the upstream boundary.

The adequacy of upstream boundaries can be evaluated with a coarse mesh early in the mesh development process. If the upstream boundary condition is set too wide, some water may flow along a path that diverges from the main channel or that takes time to distribute naturally based on the physics of the model. If the upstream boundary condition is set too narrow, high entrance velocities result near the boundary condition. This may be acceptable if changes to the width of the upstream boundary condition do not influence the results at the project area. Standard practice is to set the width of the upstream boundary condition equal to the width of a known floodplain boundary of the same discharge.

6.4.2. Downstream Boundary Sensitivity Testing

Potential problems with upstream boundary conditions are related primarily to the spatial variation of boundary conditions. Changes in downstream boundary conditions can also impact results in the project area because of the assumptions made at the downstream boundary.

To test model sensitivity of the downstream boundary condition, the downstream boundary WSE for a given discharge can be increased or decreased, and the model simulation rerun. Then the WSE results from the various model runs can be compared. This type of testing is similar to the way model sensitivity of a 1D model downstream boundary condition is evaluated (see Section 5.2.2). If no differences exist in model results in the project area when varying the downstream boundary conditions, then the model downstream limits are sufficient. If differences exist at the project location, the downstream boundary should be moved further downstream from the area of interest.

6.4.3. Model-Wide Sensitivity Testing

In addition to testing upstream and downstream boundary model sensitivity, the overall model domain can be tested for sensitivity to model-wide parameters. Just as upstream and downstream boundary conditions can influence model results around the area of interest in both 1D and 2D hydraulic models, specific model parameters can influence the results in the area of interest. For 2D models, roughness settings, geometry configurations, and internal boundary conditions can all influence model results.

Roughness values. Model roughness values have a significant influence on both 1D and 2D model results, second only to the mesh geometry. The Manning's n roughness values can be varied within an acceptable range to determine if small changes to the roughness values cause significant changes in WSE or velocities at the area of interest. Roughness values should be changed globally when performing any sensitivity testing. Roughness values for a select area of the hydraulic model should not be changed while leaving other areas unchanged without a compelling reason to do so. If minor adjustments to roughness values result in significant

changes, the selected roughness values should be reevaluated to ensure they are within an acceptable range for the roughness conditions being modeled.

Mesh resolution. Mesh resolution is another parameter that can be tested to determine its influence on the model results. Here again, there is some similarity to 1D modeling in that a more detailed cross section ground line, and more cross sections, can sometimes change 1D model results. With a 2D model, the modeler selects a mesh resolution that provides a balance between sufficient detail in the representation of the ground, appropriate resolution of hydraulic variables, and model run times. The mesh can be refined (increased element count) or coarsened (decreased element count) to provide the desired balance between mesh detail and model run times.

The modeler can refine or relax the mesh to test the sensitivity of the model results in the area of interest. The modeler may begin a model with a coarser mesh in the early phases of a project. As the mesh is refined, model results can be compared to determine if significant changes occur within the project area with respect to WSE and velocities. If model results are significantly different, the model should be refined further, and the latest two model results compared until no appreciable changes occur within the project area.

Hydraulic structures. Sensitivity testing should be conducted on hydraulic structures like culverts, bridges, and weirs since they are often primary hydraulic controls. While survey data should be collected on structures, it is helpful to the modeler to understand how model results respond to changes at structure locations. For example, if a relief culvert is blocked, how does that impact model results, or if piers are represented in more detail does that change the WSE at a bridge. These tests help modelers better understand what aspects of structures, or what specific structures, control the hydraulics.

6.5. MODEL DOCUMENTATION

During a 2D model analysis, appropriate information should be documented describing the model development, review, and results. Three steps are recommended to sufficiently document the analysis and include metadata, summary memo, and report, as shown in Figure 6.10. Each step includes similar data, but each plays an important role. In addition, it is best practice for modelers to keep a model log that includes file names, model versions, and simulation notes.

METADATA	REVIEW SUMMARY MEMO	2D MODELING REPORT
Included with hydraulic model	Provided to model reviewer	Provided with final model
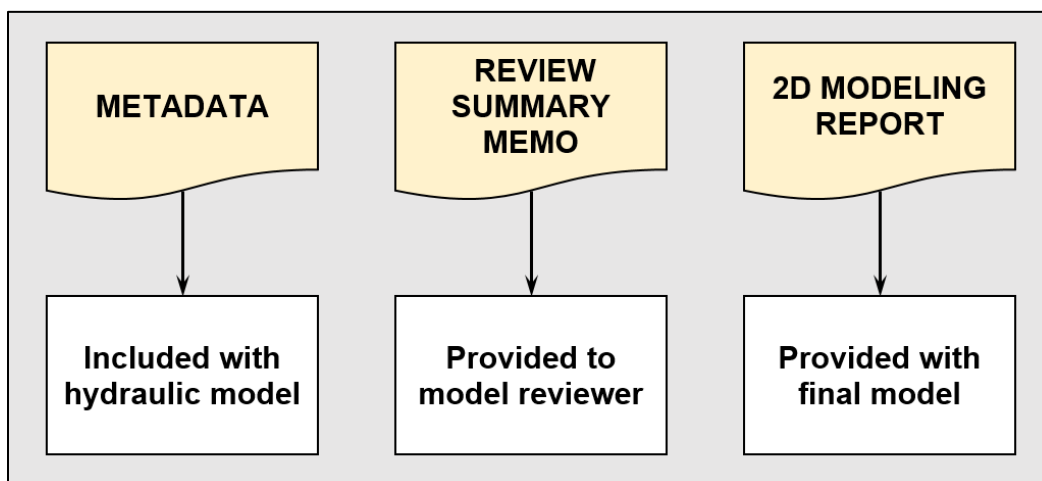
Figure 6.10. Recommended 2D modeling documentation.

Metadata is information pertaining to individual data sets. Appropriate metadata to include with the model is described in Section 4.2.3. Metadata is important because it provides information related to data sources and assumptions made during model development. Metadata is included with the model files so future users and reviewers can access the information.

In addition to metadata, the modeler should produce a model review summary. This memo should be delivered with the model to a reviewer to assist with the review process. The summary memo content should include important items to be reviewed with the model. Appendix C includes a sample review summary memo outline and model review checklist that can also assist with the review process.

Best practice is to have models reviewed by the model practitioner and another person familiar with 2D modeling, though they may or may not be familiar with the specific project. Some model applications, such as those involving floodplains, may involve a formal review due to local, state, and federal regulations.

An informal review seeks to answer the question: Do the model and results appear reasonable? The modeler should use this approach during the model development and analysis and on non-design related models. It may also be prudent to conduct an informal review by someone other than the modeler.

A formal review includes a thorough review of the model and results. This approach is recommended when a review is required for design or floodplain mapping models. This type of review seeks to answer questions including but not limited to: Was the correct data used? Is the terrain data reasonable? Is the model geometry sufficient? Are the boundary conditions correct? Is there model continuity? Is the model stable? Are hydraulic structures represented correctly? Do the results appear reasonable?

Finally, it is good practice to document the model development, analysis, calibration/sensitivity checks, and results in a 2D hydraulic modeling report. The report should include information contained in the metadata and summary memo but expand on the purpose, approach, and relevant results of the analysis.

The report content, outline, and breadth may vary depending on the application and complexity of the model. A sample 2D model outline is provided in Appendix D. This outline is intended to be used as an example for 2D modeling studies related to transportation infrastructure. However, it does provide a good basis for other 2D model applications. Modelers are responsible for determining an appropriate outline for their specific projects.

CHAPTER 7

PRESENTATION OF RESULTS

Presentation of 2D model results can provide an understanding of the physical conditions that exist in the scenario represented by the simulation (actual or proposed conditions). Two-dimensional model results of velocity and WSE also provide the basis for computation of other terms such as Froude number, bed shear stress and a wide variety of numerical quantities based in part or whole on the velocity, WSE, and depth. Plotting contours of these values provides an intuitive representation of the water surface and flow patterns over the model domain.

In addition to providing more accurate hydraulic results, 2D models offer enhanced opportunities for presentation of model results compared to 1D models. In contrast to 1D models that only provide results at specified cross sections, 2D models provide hydraulic results throughout the inundated area of the model. The intuitive and graphically rich presentation of results can be instrumental in improving project outcomes by communicating potentially complex hydraulics concepts to a broad range of stakeholders.

The most common methods for presenting results in plan, cross section, profile, and oblique (3D) views are explained in the subsequent sections. In addition to the methods mentioned above, geospatial assessment and presenting model results through animations are also discussed.

7.1. PLAN VIEW

Presentation of 2D model results in plan view offers a powerful and effective means for communicating hydraulic results to stakeholder groups that may not have the level of experience of a hydraulic engineer. Typical plan view results and standard applications are described below.

7.1.1. Water Surface Elevation and Depth

A plan view of WSE, or flow depth, allows identification of inundation limits, freeboard at bridge structures, and flow paths under various modeled flow conditions (Figure 7.1 a and Figure 7.1 b). Water surface results in plan view also provide designers with a means of assessing design parameters or risks associated with existing or proposed water crossings (e.g., roadways, bridges, and culverts). For example, the plan view of WSE may assist in:

- Evaluating the flow that causes a roadway to overtop
- Determining flow depths directly upstream of a pier for scour calculations
- Determining the freeboard at structures
- Determining the height of bank erosion protection
- Estimating when a water crossing structure is in pressure flow conditions
- Comparing WSEs to base flood elevations (BFEs)
- Assessing flooding at properties and other infrastructure adjacent to a waterway
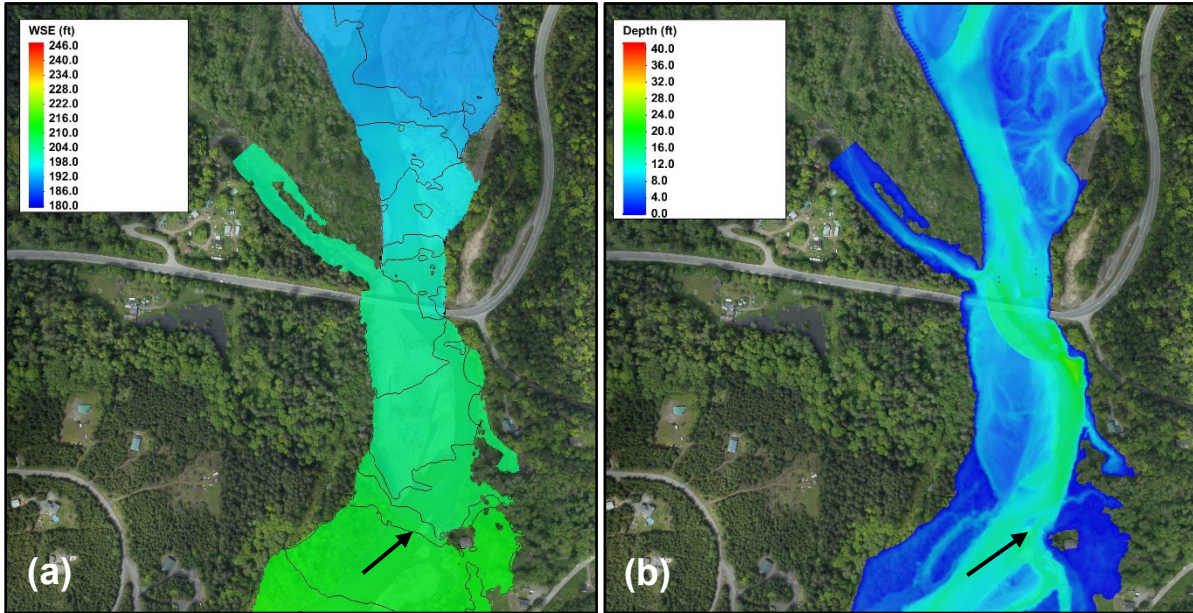
Figure 7.1. Plan view presentation of the WSE (a); and the flow depth (b) from a 2D model.

An additional benefit of 2D results is the ability to create comparison exhibits which show potential changes in WSE for different alternatives (e.g., existing versus proposed conditions). The visualization of the WSE difference is enhanced when using a two-tone coloring scheme; for example, one that employs red and blue contour shades for representing areas of predicted WSE increase and decrease.

Figure 7.2 provides an example of changes in WSE due to replacing an existing bridge with a longer span. The red (positive values) and blue (negative values) show areas of increase and decrease in WSE when compared to existing conditions, respectively. The decrease (blue) upstream of the bridge represents the lowering of backwater that resulted due to the smaller span bridge in the existing conditions. The white color depicts areas where there is no change in WSE, which can be a useful tool in assessing project impacts.
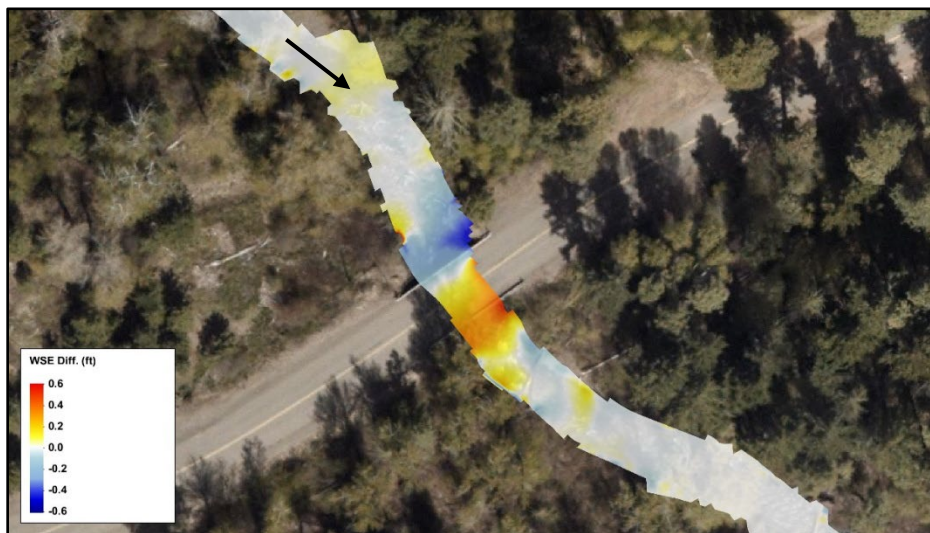
Figure 7.2. Plan view of the difference in WSE between existing and proposed conditions.

7.1.2. Velocity

Plan view velocity distributions are useful for identifying high-velocity areas, such as those developed at flow constrictions and meander bends (Figure 7.3). Additionally, plan view velocity maps may be used to highlight areas of low velocity, such as those present upstream of an undersized structure due to backwater (Figure 7.3), or in the wake of flow deflection structures (Figure 7.4). In the latter case, evaluation of the low-velocity wake extents from the plan view velocity distribution may help in determining the spacing between flow deflection structures such as barbs or spurs.

As a vector quantity, velocity directions can be represented in plan view results with vectors plotted to show the directions and magnitude of the velocities in the flow field. Vector plots can help modelers and stakeholders better understand and visualize complex flow patterns. Vectors can also be used to help modelers estimate flow directions for pier scour analysis or for other design factors where flow direction may have an impact.
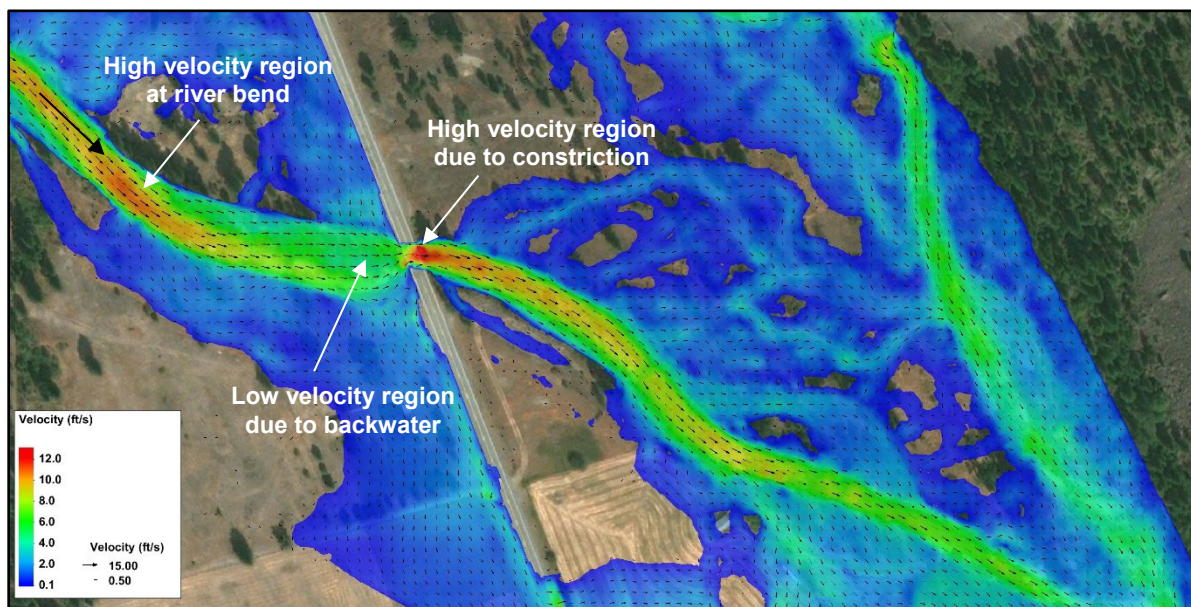
Figure 7.3. Velocity magnitude and vectors computed by a 2D model at an undersized bridge crossing.

Velocity difference plots are similar to the WSE difference plots discussed above. Results from 2D models can be used to show potential differences in the velocity plan view distribution between alternatives (e.g., existing versus proposed conditions). For example, Figure 7.5 illustrates the difference in the velocity fields with and without flow deflection structures. This figure highlights the flow deceleration near the bank (blue areas) and the increase in velocity near the channel center (red/yellow areas). When the flow deflection structures (e.g., barbs) are present, the model results show that the bank and roadway are protected during this flow simulation.

The plan view distribution of the 2D velocities is also useful for predicting scour at bridge piers. Pier scour prediction equations from HEC-18 require an input of velocity and direction located directly upstream of the pier. These parameters can be identified in the plan view velocity distributions from a 2D model. Thus, the application of a 2D model allows a user to identify the spatial variability of velocity near a hydraulic structure for use in design.
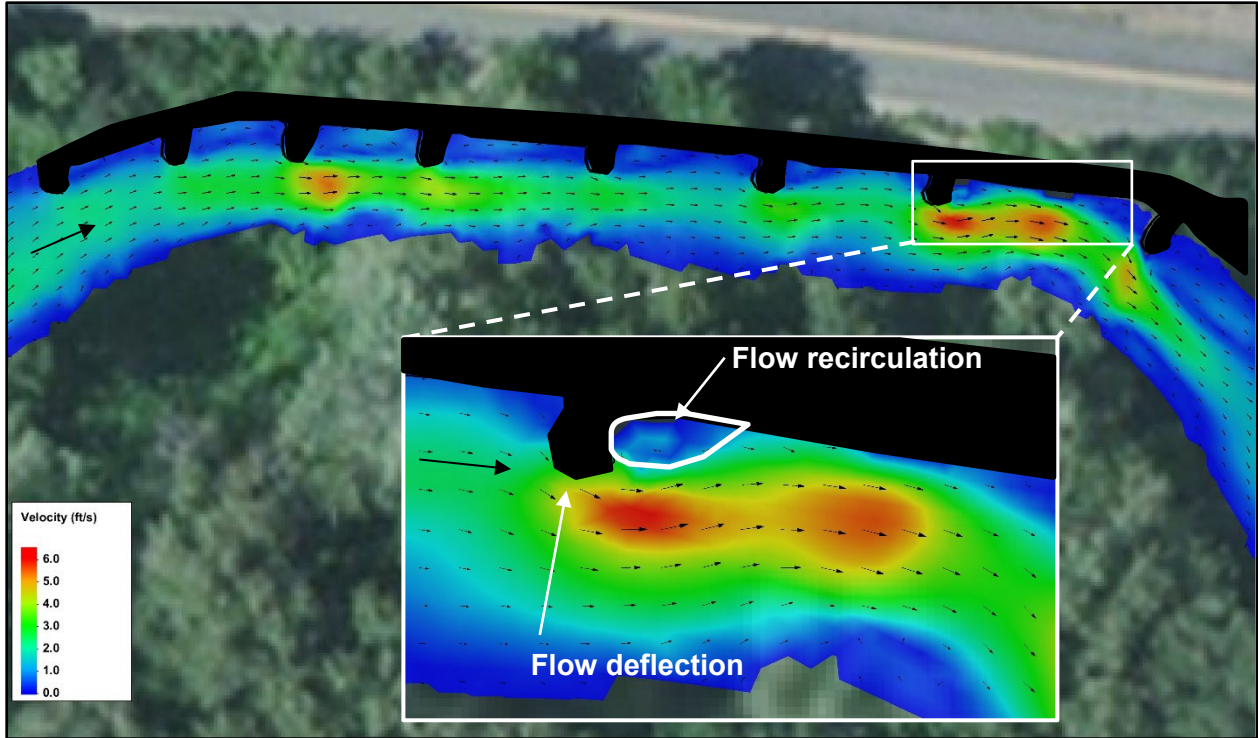
Figure 7.4. Velocity around flow deflection structures computed by a 2D model.
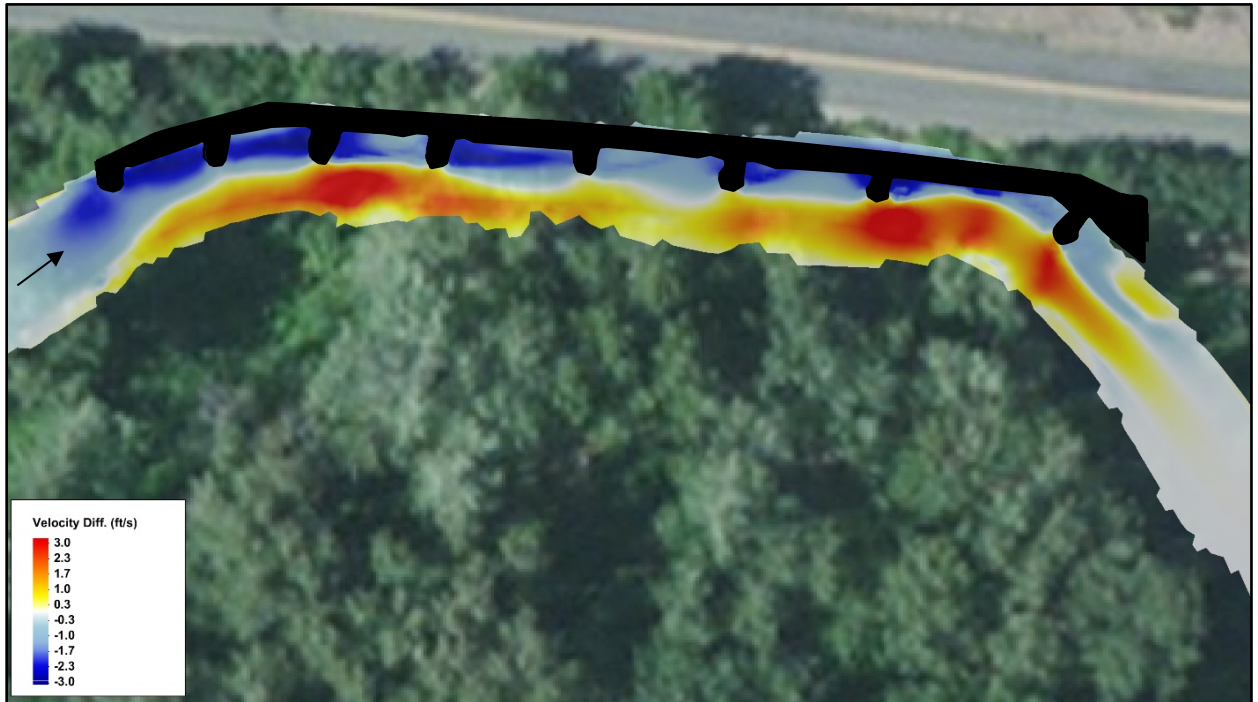
Figure 7.5. Difference in the velocity before and after the placement of flow deflection structures showing the decrease in velocity adjacent to the roadway.

7.1.3. Bed Shear Stress

A plan view distribution of bed shear stress can identify regions of high and low shear (Figure 7.6), which can assist in identifying areas prone to sediment erosion and deposition.

The plan view distribution of bed shear stress may also be useful in the design of bank protection and in scour calculations. For example, the maximum bed shear stress on rock to be used for bank protection can be determined from the plan view and can also be used to determine the minimum size of rock that remains stable following design guidelines provided in HEC-23. In addition, extraction of the bed shear stress values around piers and abutments derived from 2D model output may be used to identify regions with a high potential for scour (HEC-18).
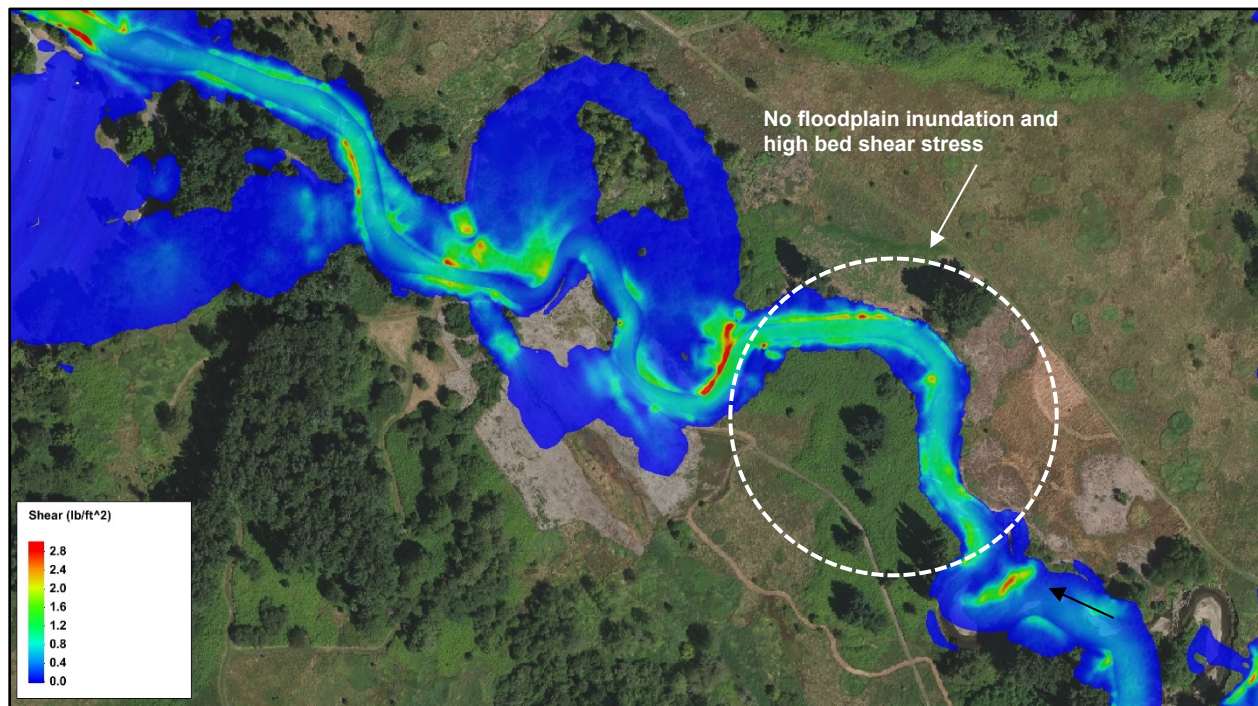
Figure 7.6. Plan view distribution of bed shear stress calculated by a 2D model.

7.1.4. Froude Number

Froude number results can help assess if the appropriate range of Manning's roughness values were used. If supercritical flow exists along a significant portion of a reach, it may indicate n-values used are too low, and a re-evaluation of roughness coefficients may show that not all energy losses were accounted for (Jarrett 1985). In addition to assessing the Froude number to assist in roughness estimation, the distribution of the Froude number can also be presented in plan view to show locations of severe contractions. In Figure 7.7, the 2D model predicts supercritical flow regions directly downstream of a concrete arch bridge that experiences pressure flow during the modeled 100-year event. This flow pattern is often observed at flow constrictions or rapid changes in bed topography. Conversely, the figure shows subcritical flow regions directly upstream of the bridge due to the backwater caused by the existing bridge.
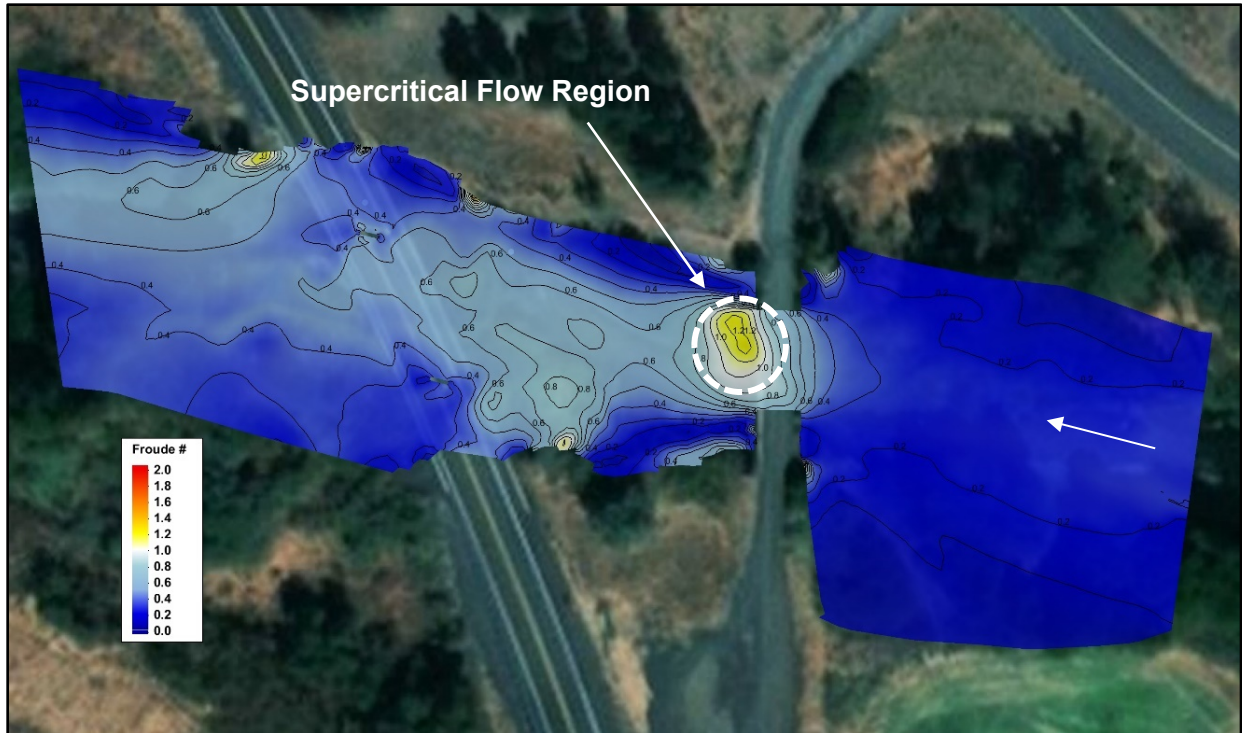
Figure 7.7. Plan view distribution of the Froude number calculated by a 2D model.

7.2. CROSS SECTIONS AND PROFILES

Cross section and profile views are common methods for viewing hydraulic results due to the extensive use of HEC-RAS. Typical cross section and profile view results for water surface elevation, depth, velocity, and bed shear stress are described in the sections below.

7.2.1. Water Surface Elevation and Depth

Cross Section. Cross section displays are useful for illustrating WSE and depth computed by 2D models. Figure 7.8 illustrates the variations in hydraulic values, specifically WSE and depth in the vicinity of a bridge. In this example, the results are plotted just upstream of a 3-span bridge where, under existing conditions, the piers have a large effect on WSE. In addition to the 2D model results (blue line), which vary along the cross section, an averaged value, similar to what a 1D model may provide, is shown with the red dashed line. Using the averaged (1D) value could have large implications in the determination of freeboard for a bridge, calculation of local scour, or design of a scour countermeasure. Therefore, a modeler should carefully choose the most applicable model for the project site to be used in the design of safe transportation infrastructure.

Although local scour equations need velocities in the vicinity of the pier, contraction scour equations require the use of cross sections to extract the average flow depths from a 2D model. Average flow depths are used both for determining whether the flow upstream of the bridge is transporting material (e.g., live-bed or clear-water) and for the clear-water contraction scour (HEC-18).
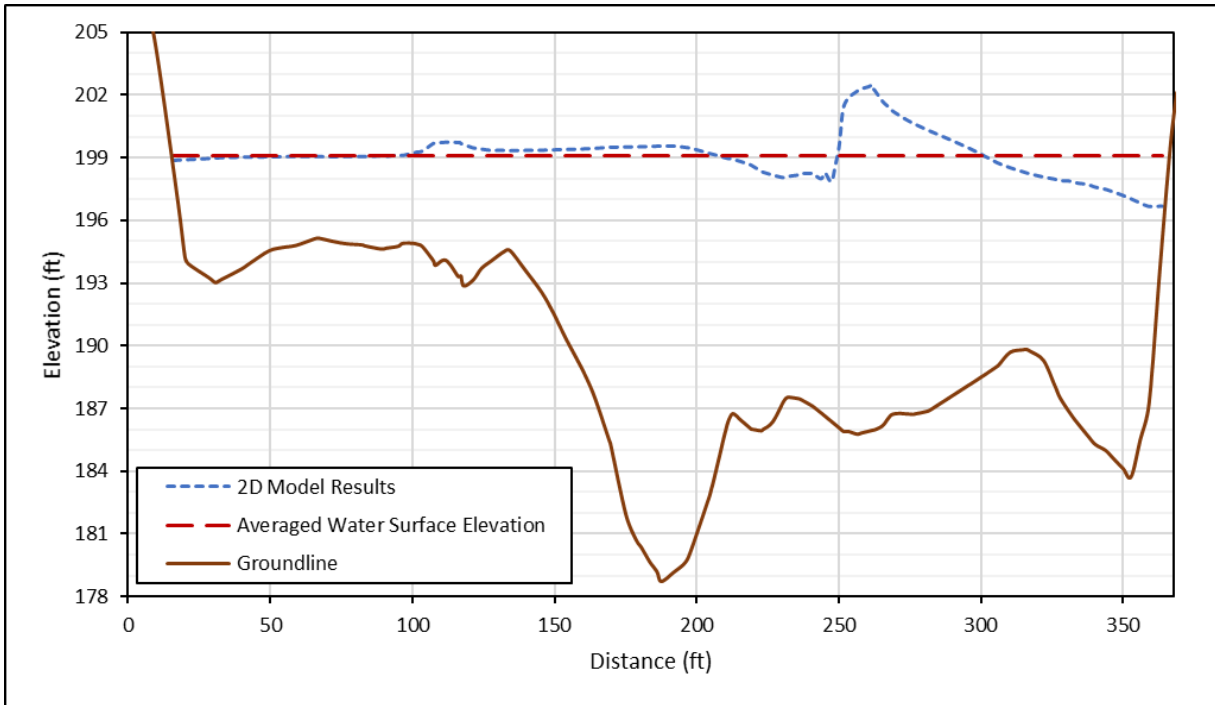
Figure 7.8. Cross section plot of 2D and average WSE.

Channel Profile. To assess gradually varied flow along a channel profile, WSE values can be extracted from the results of a 2D model and plotted together with the ground elevation values along a user-determined line. The WSE profile can clearly show the presence or absence of backwater upstream of a structure, as well as the longitudinal extent of the backwater. For example, Figure 4.7 and Figure 7.9 shows the WSE profile extracted along a user-defined river centerline near a bridge. In Figure 7.9, the existing bridge located near station 1,600 feet introduces backwater that extends for about 500 feet upstream of the structure and causes approximately a three-foot increase in the WSE at the upstream face of the existing bridge. This type of assessment can be used to assist in sizing bridges (Section 8.1.1) to meet design and regulatory requirements.
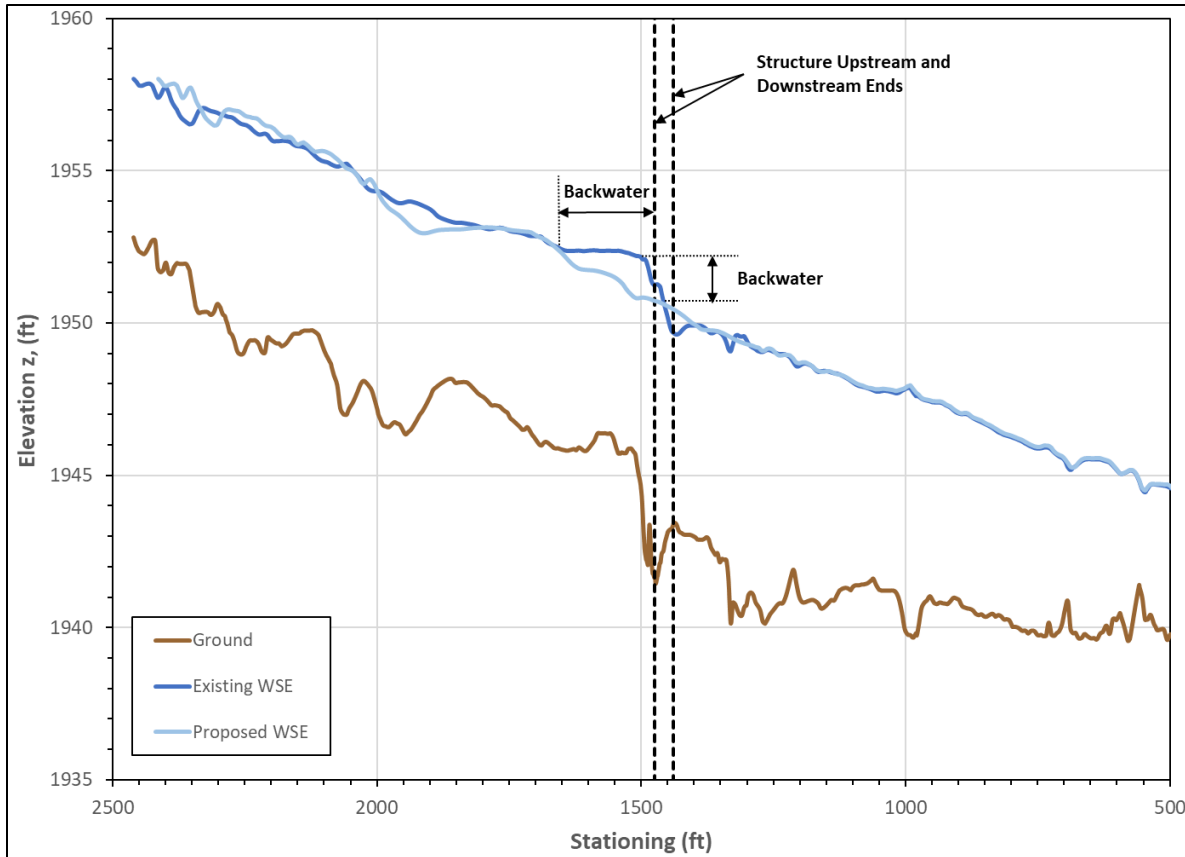
Figure 7.9. Water surface and ground profiles from a 2D model.

7.2.2. Velocity

Figure 7.10 illustrates the variations in velocity values in the vicinity of a bridge. In this example, the results are plotted just upstream of a 3-span bridge, where under existing conditions the piers have a large effect on the velocity. The velocities are reduced in the immediate vicinity of the two in-water piers (located at approximately Stations 110 and 255) and increases between the piers due to the flow contraction through each span.

In addition to the 2D model results (blue line), which vary along the cross section an averaged value, similar to what a 1D model may provide, is shown with the red dashed line. Using the averaged (1D) value could have large implications on the calculation of scour (HEC-18) or the design of a scour countermeasure (HEC-23). If the averaged (1D) velocity model result was used in Figure 7.10, it may underpredict the actual velocity by up to a factor of two (~16 feet per second compared to ~8.5 feet per second). In this regard, 2D model results often provide more accurate information than traditional 1D models for the design of safe transportation infrastructure.

7.8
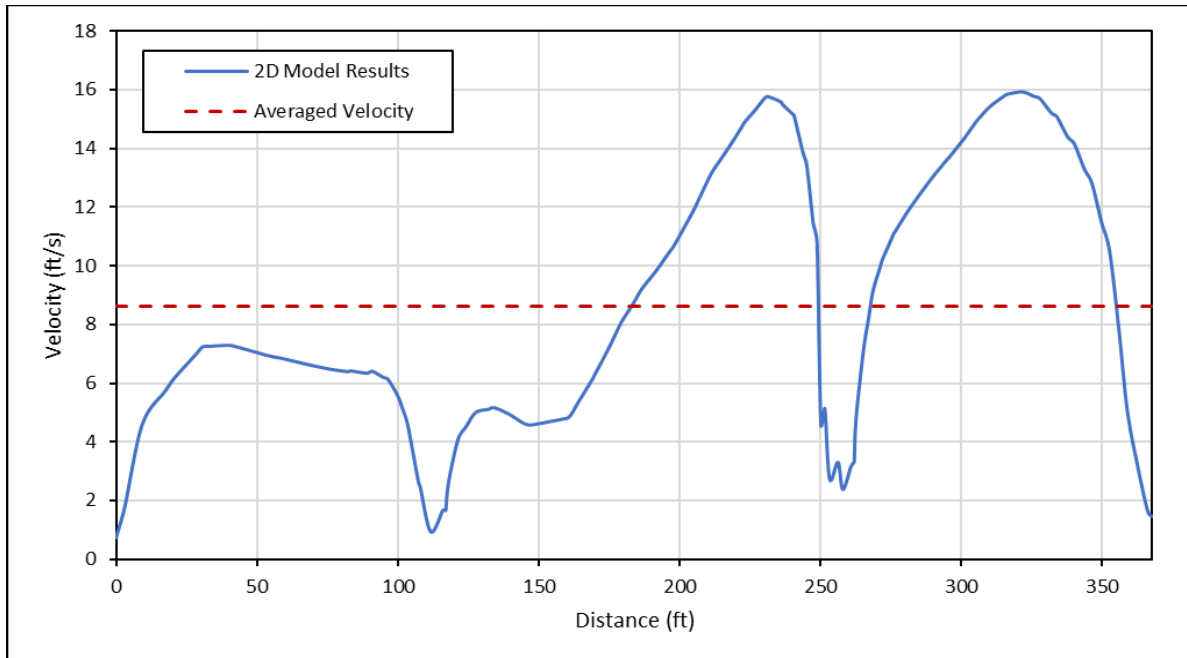
Figure 7.10. Cross section plot of 2D and average velocity.

7.2.3. Bed Shear Stress

The bed shear stress along a cross section can be used to calculate the cross-sectional average of bed shear stress, which can be used in sediment transport prediction formulas, along with sediment and hydraulic information, for predicting the sediment transport capacity. Furthermore, the bed shear stress along cross sections may be used as an alternative to the plan view bed shear stress distribution for predicting abutment scour (HEC-18) and designing bank protection. In this case, the maximum bed shear stress along the cross-section at the location of the structure may be used in the appropriate equations.

7.3.   OBLIQUE VIEWS

In addition to plan, profile and cross section views, oblique views can provide a powerful and effective means for communicating hydraulic results to stakeholder groups that may not have the level of experience of a hydraulic engineer. Figure 7.11 illustrates an oblique view of ground elevations with the 100-year WSE inundation limits shown in transparent blue. Figure 7.12 provides an example of a bridge in 3D and how oblique views can be used to review how well the mesh represents the terrain and verify that it is consistent with the original terrain surface. In addition to still images depicting hydraulic results, oblique view model results can be exported as videos, which are further described in Section 7.5.
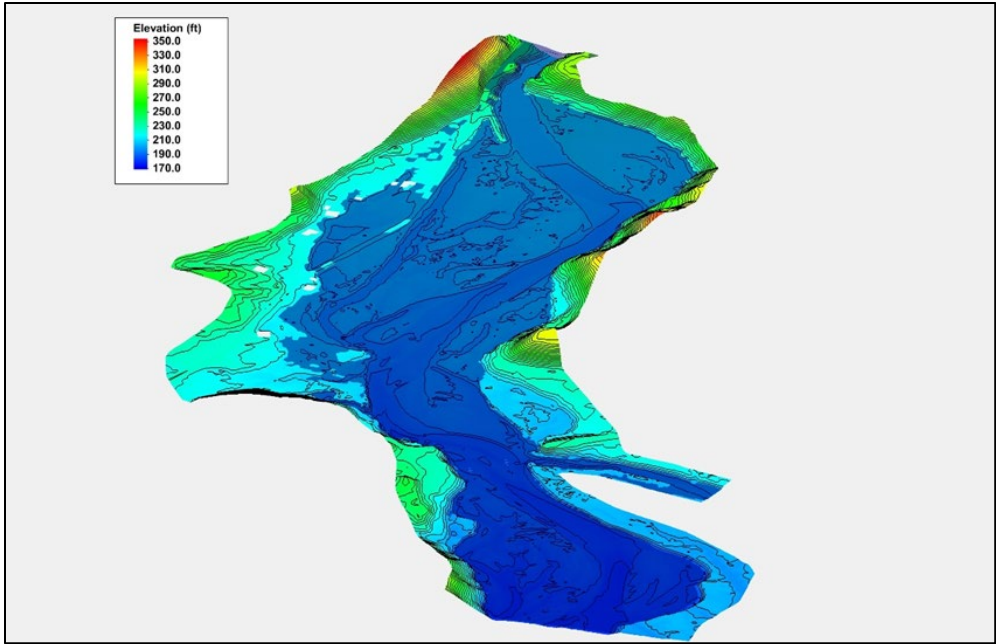
Figure 7.11. Oblique view of ground elevations with the 100-year water surface extents.
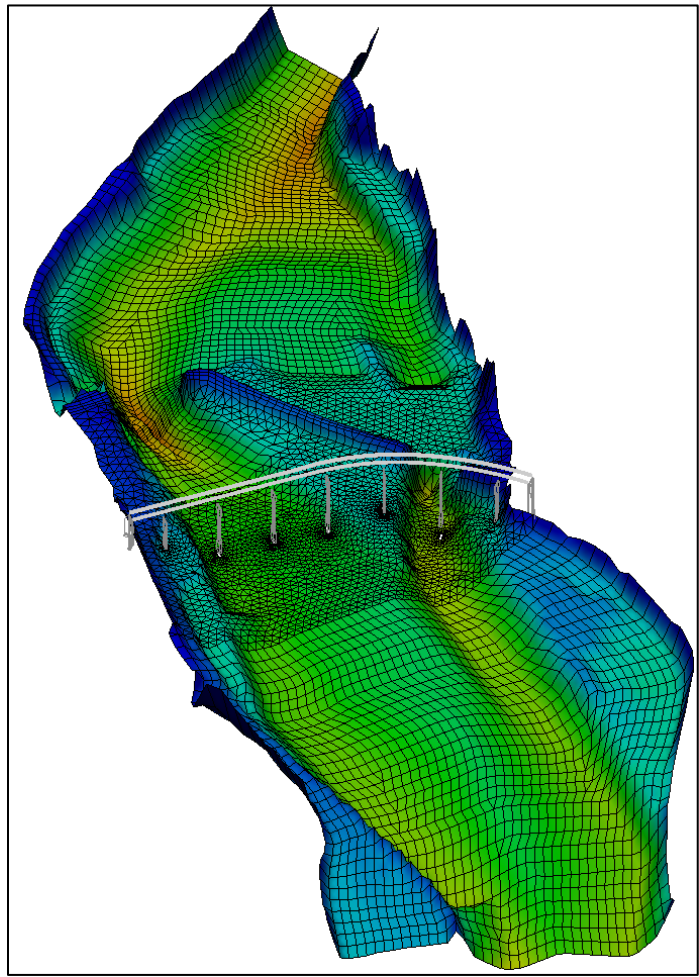
Figure 7.12. Oblique view of mesh geometry and colored elevations at a bridge crossing.

7.10

7.4.   GEOSPATIAL ANALYSIS

Two-dimensional model results provide the spatial distribution of variables, such as WSE and velocity, by calculating their values at every computational point in the domain and interpolating between points. The resulting spatial distribution of the WSE and velocity are useful in various applications, such as for illustrating the extents of a floodplain or the flow patterns around a hydraulic structure (see Section 7.1). The spatially variable distributions of the WSE and velocity calculated by numerical models allow the user to perform various geospatial analyses, including:

- Calculation of other hydraulic engineering parameters
- Simulation comparisons
- Data conversion for other software
- Zonal classification

The aforementioned types of geospatial analyses permit additional insight into complex hydraulic flow patterns, which may be beneficial for certain engineering and geomorphic applications as well as for communicating results to stakeholders. These geospatial analysis types are discussed in more detail below.

7.4.1. Calculation of Other Hydraulic Parameters

Most 2D models have the capability to perform algebraic, geometric, and temporal operations on the computed WSE and velocity data sets to create new data sets. In its simplest form, algebraic operations of the calculated data sets include the computation of flow depth, as the difference between the water surface and ground elevation. This difference, which is performed at each computational element in the domain, results in a flow depth data set.

In practice, many of the available models perform preliminary extra computations during runtime to compute and output additional data sets of this type, including flow depth, bed shear stress, and Froude number. More complex algebraic operations are also possible in most 2D models, such as the calculation of unit discharge and unit stream power. The computation of the unit discharge and stream power are examined in additional depth in the subsequent subsections.

In addition to algebraic operations, new data sets can be computed based on geometric (spatial) analysis of values to derive slopes and gradients of the data sets (i.e., how the values change in space). In addition, temporal analyses can be performed on unsteady simulations to evaluate the temporal derivative of a data set, which may show how the spatial distribution of the computed data set changes through time.

Unit Discharge (Depth-Velocity Product). The unit flow rate (q) data set can be computed simply by taking the product of the velocity and depth data sets (q = VH). Calculation of the unit flow rate data set (Figure 7.13) has several applications, including:

- Computing and visually assessing flow distribution at a cross-section, especially at bridges
- Simplifying the computation of critical flow velocity
- Determining effective flow corridors for floodway determination
- Evaluation of potential damage in flood mitigation planning
- Computing parameters for abutment scour calculations
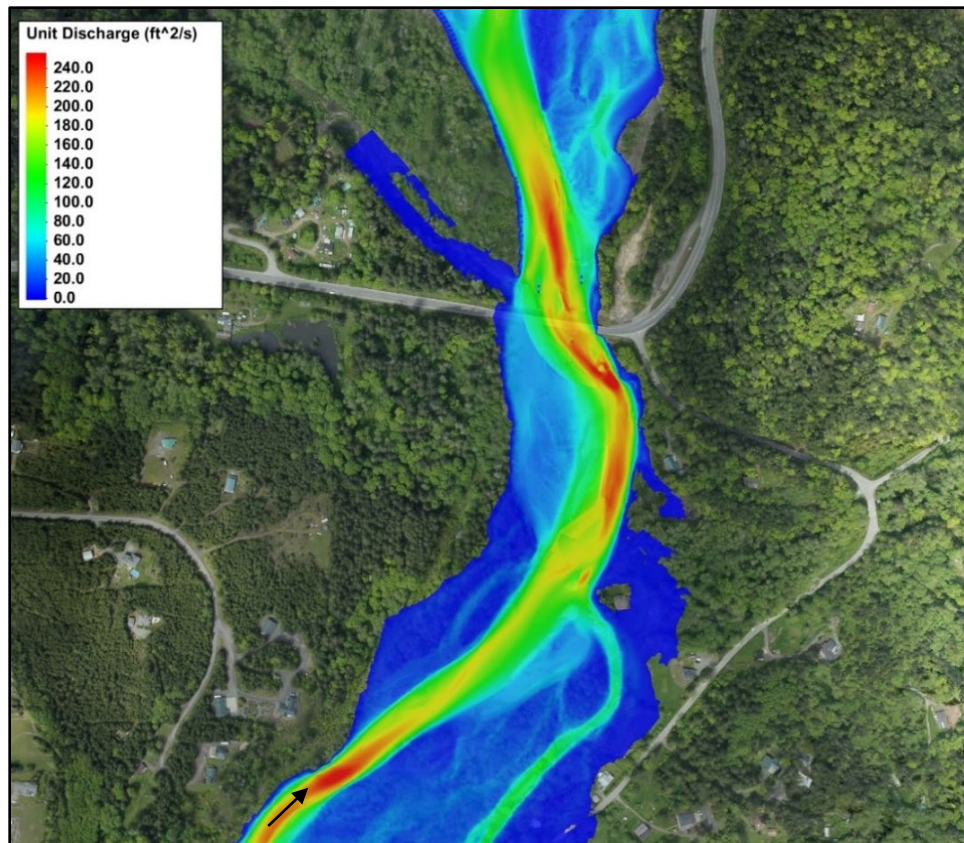
Figure 7.13. Example of the unit discharge at a bridge crossing.

Unit Stream Power. Stream power is a hydraulic variable that expresses the rate at which a river is expending energy due to the energy dissipation against its bed and banks. Most geomorphic analyses consider stream power per unit area of the river, which is the product of the unit weight of water, unit discharge, and slope of the energy grade line. Alternately, the unit stream power can be computed by multiplying the shear stress and velocity (HEC-18). The latter definition may be used to estimate unit stream power from the results of a 2D model, by multiplying the shear stress and velocity data sets, both of which are typically computed by 2D models (Figure 7.14).

Stream power may be used for assessing the type of bedforms at the modeled reach utilizing the regime diagram (Simons and Richardson 1961), and for predicting river channel patterns (van den Berg 1995). Furthermore, the unit stream power may be used for predicting sediment transport rates, utilizing unit stream power-based sediment transport prediction formulas, such as those of (Bagnold 1980), and for predicting scour in erodible rock following the approach outlined in Section 7.13 of HEC-18.

While the analyses mentioned above use the stream power as derived by the multiplication of the bed shear stress and velocity data sets of a 2D model, other formulations and geomorphic analyses may use alternate definitions of stream power. For instance, the Yang (1972) sediment transport rate prediction formula requires input of the stream power per unit water volume, which requires dividing the stream power per unit area with the flow depth. Therefore, when conducting a geomorphic analysis that involves calculating the stream power from 2D model results, it is recommended that the user verifies the definition and units of the stream power used in the desired analysis and adjusts the data set calculations accordingly.
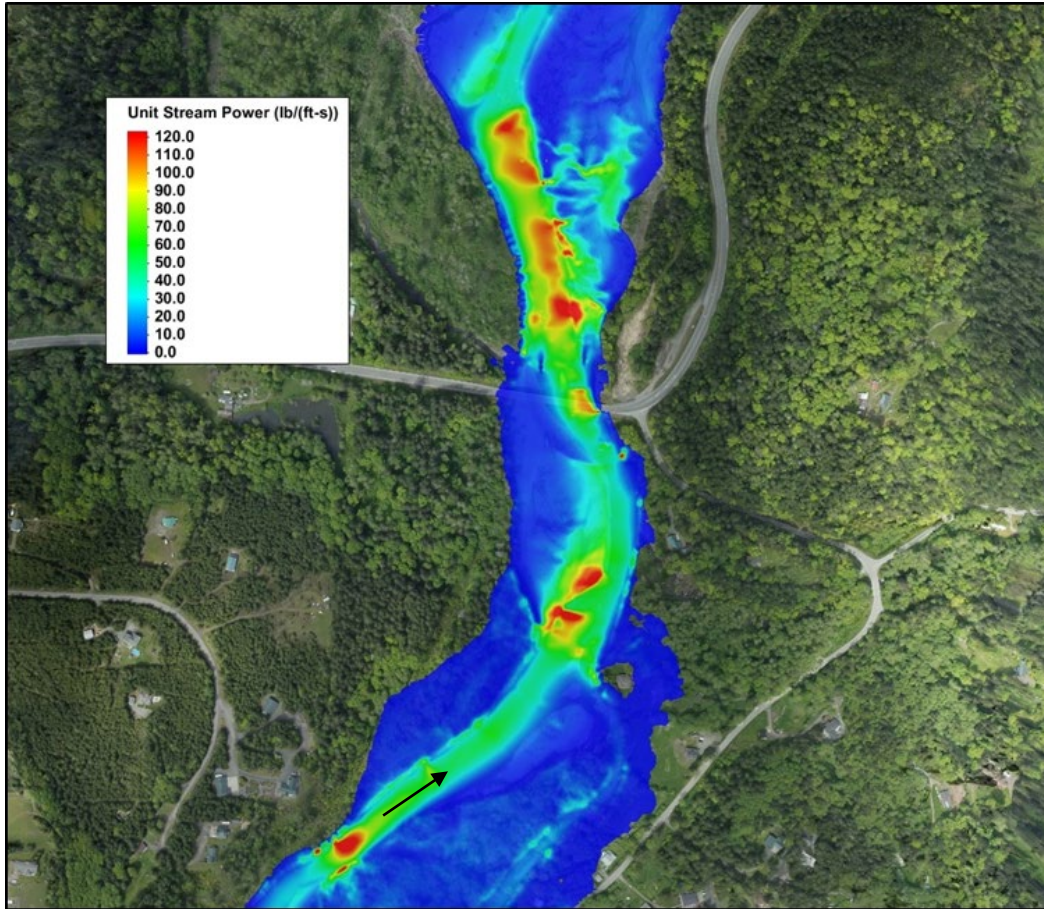
Figure 7.14. Example of the unit stream power along a river.

7.4.2. Simulation Comparisons

Beyond the visualization of a single simulation, the results can be used to compare two simulations, as discussed previously (see Figure 7.2 and Figure 7.5). For example, one simulation may represent existing conditions in a study area, while a second simulation could represent proposed conditions after the construction of some new features, such as a bridge or other modification of a river geometry. Two-dimensional models typically have functionalities that allow computing the difference in the values of a computed hydraulic variable (e.g., WSE or velocity) at corresponding spatial locations of the existing and proposed conditions simulations. This results in a new data set that quantifies the effects of the proposed modification on the magnitude of a computed hydraulic variable. Most common applications of the multiple simulation comparisons involve comparing the WSE at a river crossing before and after the construction of a new or replacement bridge (Figure 7.2) and comparing the flow velocity magnitude and patterns around river training structures (Figure 7.5).

7.4.3. Data Conversion for Other Software

Some modeling programs may not include all the features and capabilities discussed above, but they should be capable of exporting the 2D model results in formats that can be used for further analysis in other software packages. Commonly available software packages that are typically used for supplemental data analyses include:

- GIS applications, such as ArcView, ArcMap and Global Mapper (GIS applications commonly support several formats, but rarely support model-specific formats)
- CAD programs, such as AutoCAD or MicroStation
- Visualization programs, such as Paraview and TecPlot

Conversion from model-specific formats into a standard file format, such as a digital elevation format (i.e., xml, tif, asc, etc.) or a point files (i.e., xml, las, csv, etc.), also facilitate the import and use of the data. Some of the available graphical interfaces available today support these types of conversion.

7.4.4. Zonal Classification

In addition to the direct interpretation of the data sets created by a 2D model and the data sets derived from the results, additional insight can be gained by spatially classifying data sets in relation to specified criteria, referred to as zonal classification. The criteria can be based on data set values (ranges) and material properties or other attributes specified for polygonal regions, as is common with GIS products. Zonal classification may, for instance, be used to delineate regions of a study area that are prone to erosion or deposition. In this case, the zonal classification is used to isolate the modeling domain regions, where the computed bed shear stress is larger or smaller than the critical bed shear stress for incipient motion of the riverbed material. The area in Figure 7.15 shown indicates where the bed shear stress is larger than the critical bed shear stress, thus indicating erosion for the specified flow event.

A zone may be defined by one or more criteria. When multiple criteria are specified, zonal classification tools create polygons that bound the intersections or unions of polygons for regions satisfying the specified criterion. For example, zonal classification can be used for identifying areas of a river that may support spawning of a specific species of fish. In this case, zonal classification can be used to highlight areas of the domain where depth is above a minimum threshold and velocity is below a maximum threshold.
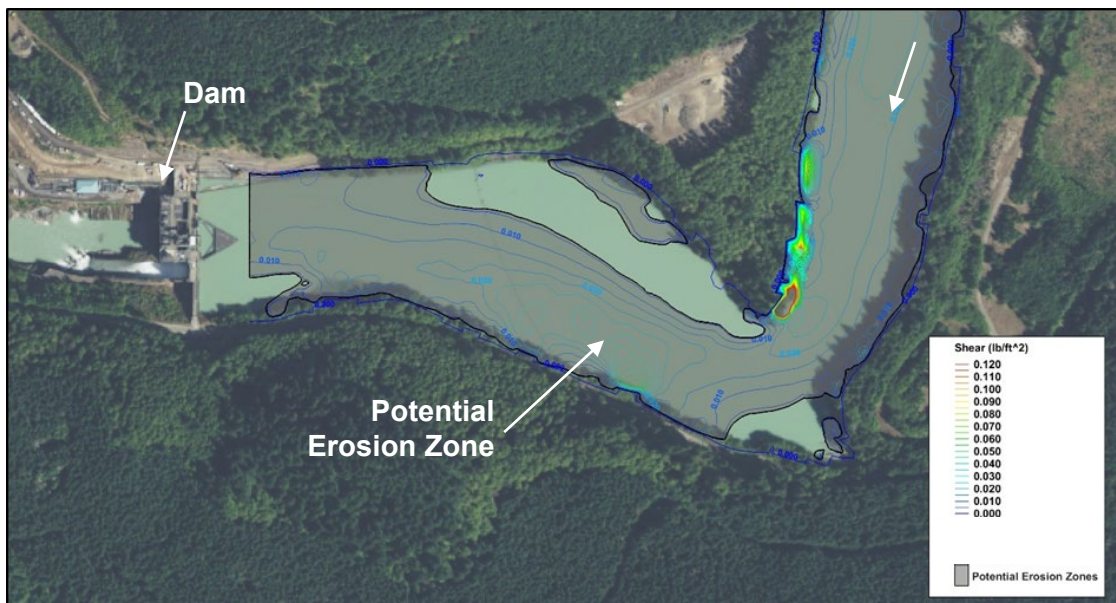
Figure 7.15. Areas prone to erosion (shaded) where the calculated bed shear stress is greater than the critical shear stress for incipient motion.

7.5. ANIMATIONS

Hydraulic analysis results can represent a field in motion as water flows through the domain. In the case of hydrodynamics, not only is the water flowing, but the nature of that flow is changing through time. Another benefit of 2D models is that results can be presented with animations or movies, which can facilitate the intuitive understanding of the results. A variety of options that most models are capable of producing are described below.

Transient or Time Step Animations. Generating a series of images that increment through the time range represented by a 2D simulation creates a transient animation product derived from a 2D solution. This requires transient data and can include either contours, vector plots, or a combination of both.

Flow Trace and Drogue Animations. Even though a steady state or single time step solution does not change through time, it still represents a flow field that can be visualized through an animation. The physical equivalent of this varies from the childhood game of stick racing in a stream to scientific studies that place tracers, such as a dye, in a flow field (Figure 7.16). Observing the motion of the sticks or dye in the field provides insight into the flow patterns. For example, locations where objects move more quickly are indicative of faster currents and locations where objects become trapped, suggest the presence of eddy currents. In numerical results, this field can be displayed by using a "flow trace" or "velocity trace" capability.

These approaches introduce a numeric "tracer/particle" and compute the path that particle follows through the flow field. The animation results from generating multiple frames of motion. Often in these types of animation, the motion can be exaggerated to result in a more dramatic visual image of the flow. Alternatively, when the motion of the results represents the actual physical speed of the flow field, the animation is referred to as a "drogue plot" animation.
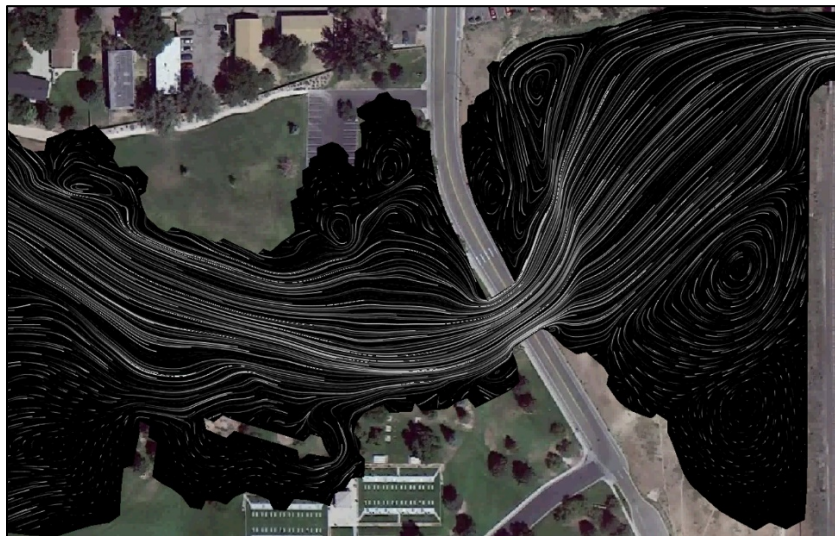
Figure 7.16. Example of a still image captured from a flow tracing video.

Observation Profile Plot Animations. In addition to the visualization of the 2D flow field in plan view, the model output can also be used to visualize local extractions along profiles or cross sections. This results in a 2D plot at a specified line through the model domain. When the model results include transient data, the 2D plot can be animated to represent the variation of the specified variables through time.

CHAPTER 8

MODELING APPLICATION

Two-dimensional models are useful tools for many transportation hydraulic engineering applications, including, but not limited to, the design of waterway crossing structures such as bridges and culverts; bridge scour analyses; bank stabilization and scour countermeasure design; floodplain analyses; and river rehabilitation and aquatic habitat assessment. The following sections provide a further discussion of these applications.

8.1. ROAD CROSSINGS

Roadway encroachments onto floodplains present hydraulic modeling, design, and regulatory compliance challenges. See, e.g., 23 CFR, Chapter I, Subchapter G (Bridges, Structures, and Hydraulics); and 44 CFR, Chapter I, Subchapter B (Insurance and Hazard Mitigation, National Flood Insurance Program). A roadway encroachment (either a bridge or culvert crossing of a river and floodplain or a longitudinal roadway encroachment) presents a constructed hard point in the river system and can introduce changes to the flow field, including:

- Increased velocities and shear stresses on the channel bed, banks, and structures within the floodplain
- Increased WSE upstream of any encroachment
- Local turbulence near obstruction
- Lateral components to flow

These changes often affect the sediment transport capacity of the system during typical and flood flow conditions. Encroachments can substantially alter the river's natural interactions with its bed, banks, and floodplain, affecting sediment transport and scour, which is most commonly evaluated at three time and spatial scales:

- Geomorphic effects, impacting the lateral and vertical dynamic equilibrium of the river system
- Reach-scale effects, including contraction scour
- Local-scale effects, including flow separation and local scour (abutment, wall, and pier)

The most common application to assess reach and local-scale effects is with the use of a rigid-bed hydraulic model. A rigid-bed hydraulic model does not update the terrain of the model based on sediment transport or scour around a hydraulic structure. A rigid-bed hydraulic model uses the hydraulic results with empirical equations available to compute reach-scale (e.g., contraction scour) and local-scale (e.g. local scour) parameters to be used in design (HEC-18). In contrast, a mobile bed model uses sediment transport relationships to estimate changes in terrain through a specified hydrograph and updates the model domain at a specified time step. Mobile-bed models are briefly explained in Section 8.4.

Encroachments can also impact aquatic organism passage (AOP) and aquatic/riverine habitat conditions. Design of a roadway encroachment onto a floodplain should:

- Evaluate hydraulic, channel stability and scour, and habitat effects
- Protect the traveling public, develop cost-effective infrastructure investments, and protect the environment
- Mitigate undesirable adverse impacts if necessary

Two-dimensional modeling of highway encroachments can provide improved hydraulic insight into encroachment design problems and consequently provide opportunities for greater design flexibility and optimization in an increasingly constrained design environment. The following sections present 2D modeling applications to various roadway crossing design and analysis problems.

8.1.1. Bridge Sizing and Approaches

Typical components for the design of a safe bridge influenced by hydraulics include the location, span, elevation, and foundation depth. Assessment of the required location and span for a bridge are generally based on the velocity, width, and direction of a design flow, both upstream and downstream of a bridge site. Two-dimensional models provide significant improvements in calculating hydraulic variables to support sizing bridges and approaches, compared to 1D models (HDS7), such as:

- Flow distribution
- Velocity distribution
- Velocity magnitude
- Velocity direction

- WSE
- Flow depth
- Shear stress
- Backwater

This is mainly due to 2D models having less assumptions than 1D models, especially for complex, natural river channels. Therefore, FHWA has a preference for the use of 2D models over 1D models for complex waterway and complex bridge hydraulic analyses (HDS7). The NCHRP project *Criteria for Selecting Hydraulic Models* (NCHRP 2006), provides some recommendations for when a 2D model may be more applicable. Some examples include bridge projects with:

- Multiple openings
- Wide floodplains
- Skewed roadway/bridge alignments
- Road overtopping

- Bends, confluences, and angle of attack
- Multiple channels
- Tidal conditions
- Countermeasure design

A multitude of geomorphic conditions exist in rivers that introduce complex flows, especially near bridges. Some of these conditions include when a bridge is at or near a river bend or meander, near a river channel expansion or contraction, has in-water piers, or when the bridge piers or the bridge itself are skewed relative to the flow direction. At these locations, the WSE and velocities are typically not constant due to the lateral flow component (see Figure 7.8). As mentioned in previous sections, 2D models calculate lateral variations in WSE and velocity, while 1D models calculate a constant WSE or velocity along an assumed cross section. Because of this, 2D models directly incorporate the effects of items bulleted above (e.g., in-water piers, bridge skew, etc.), as well as expansion and contraction. This is beyond the capability of 1D models, which use empirical coefficients to evaluate energy losses to determine hydraulic variables used to size bridges and locate approaches. These coefficients introduce uncertainty into the 1D model WSE and velocity estimates, particularly in complex flow situations.

Figure 8.1 shows an example of some of the benefits of a 2D model, where the velocity magnitudes (length of the arrows), velocity direction (direction of arrows) and WSE (black contour lines) are not uniform through the bridge reach as would be the case if a 1D model was used. This information is critical to sizing and locating a bridge superstructure and substructure (e.g., abutments, piers, etc.) elements in addition to designing proper scour countermeasures.

A 1D model would likely underestimate the WSE and velocity magnitudes to be used for design. In addition to WSE and velocity magnitudes, the direction of flow (velocity vectors) is critical in determining the proper angle of attack on a bridge pier for computing a bridge scour analysis and determining a proper bridge span. As mentioned previously, with a 1D model, the modeler assumes the direction of flow (location and alignment of cross section), which can be difficult on a bridge similar to the one shown in Figure 8.1. If the modeler assumes the incorrect direction of flow (skew) through the bridge opening it may result in underpredicting the amount of scour at an in-water pier or abutment.
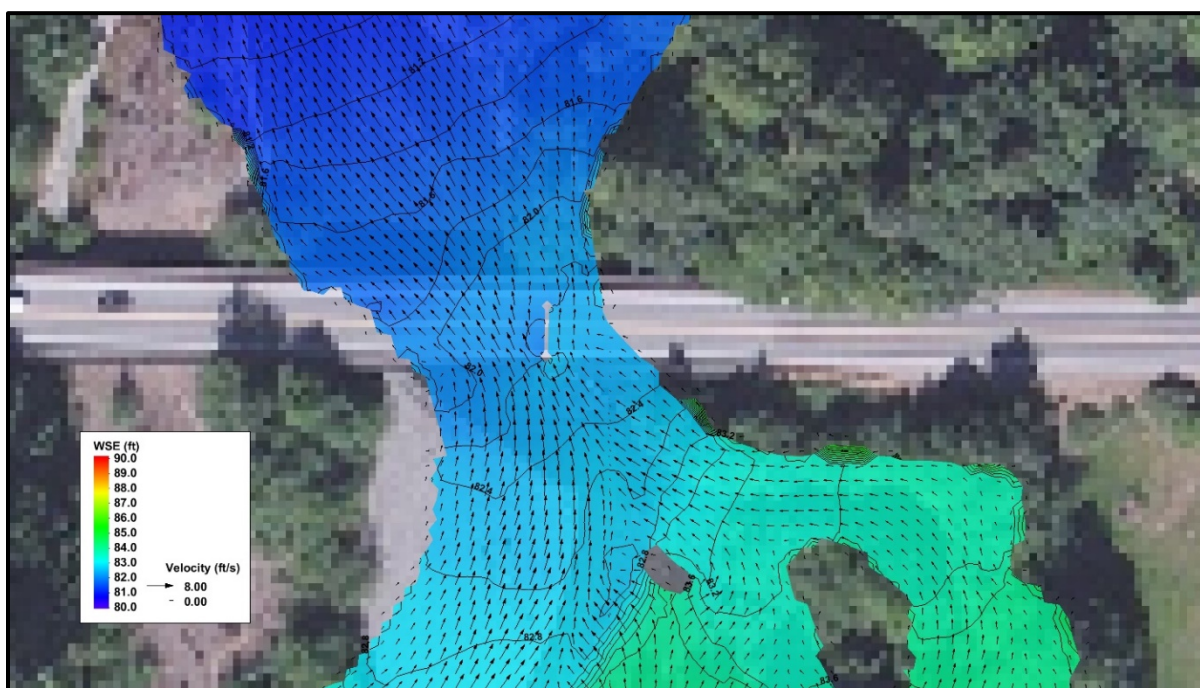
Figure 8.1. Velocity magnitudes and directions through a bridge crossing with in-water piers.

Because of the inherent benefits, discussed in more detail below, 2D models also provide a more reliable prediction of the freeboard and water surface width for a design flow, thus providing more reliable estimates for the required location, span, and elevation for bridges and their approaches.

Freeboard Estimation. In general, freeboard is the difference between the highest WSE and the lowest elevation of a waterway crossing structure. The calculation of freeboard can vary in the streamwise (Figure 7.9) and transverse (Figure 8.2) directions based on where the WSE is selected. Using cross section, profile, and plan views together (Figure 4.7, Figure 7.9 and Figure 8.2) allows the designer to select the most appropriate WSE for the determination of bridge low chord elevations and thus freeboard.

Figure 8.2 depicts a cross section of WSE values (blue line) calculated by a 2D model at a bridge crossing. This figure also shows the average WSE (blue dotted line), the ground elevation (brown line), and the bridge low chord elevation (black dotted line). The 2D model results capture the variability of the WSE along the cross section, which allows a designer to evaluate how the WSE and freeboard, varies along the cross section. In this example, the maximum freeboard is approximately three feet at Station 48, near the right bridge abutment. Moving toward the left bridge abutment, the freeboard decreases, reaching a minimum value of one foot at Station 22. 2D models provide a more definitive estimate of the freeboard than is possible with 1D models, which cannot capture the transverse and longitudinal WSE variability. A 1D model would predict

a constant WSE value as shown (blue dotted line), thereby predicting the same freeboard (~1.3 feet) along the entire cross section and potentially underestimating the correct WSE and freeboard to be used in the design of the bridge.
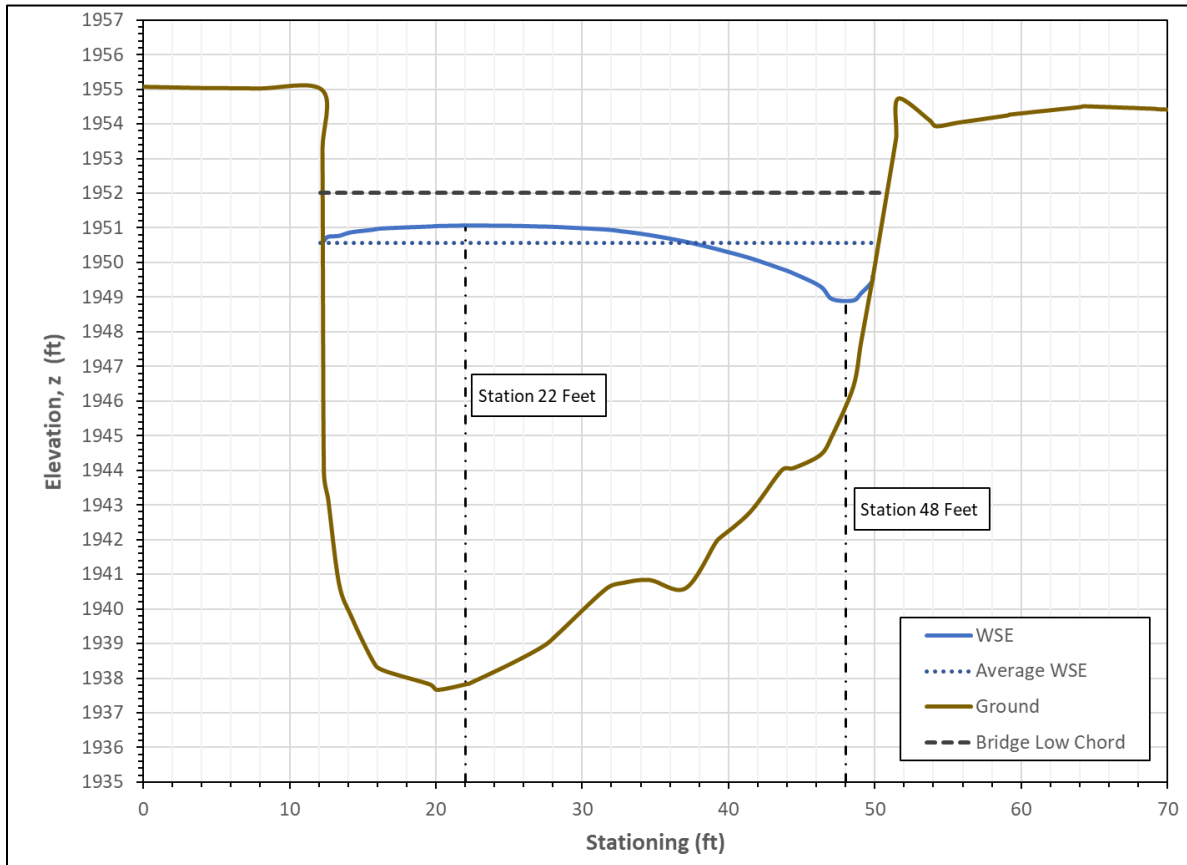
Figure 8.2. Cross section of the water surface and ground elevation at a bridge crossing.

For design purposes, a common approach is to assess the WSE approximately one bridge with upstream of the face of the bridge for determining the design WSE and the reference for computing freeboard. In many cases, the rapid draw down and precise profile of the WSE's through the bridge opening are difficult to predict. Consequently, it is more reliable to reference the approach WSE, upstream of the bridge.

Another approach for where to assess WSE for determining freeboard is illustrated in Figure 4.7 and Figure 7.9. The profiles in these figures provide the designer with the location where the water surface slope changes compared to the adjacent upstream reach. The location of where a sudden change in water surface slope is likely tied to the influence of the crossing and may be a good location for selecting the WSE for use in freeboard determination. In addition, the plan view in Figure 4.7 shows WSE contours for systems where the floodplain extents are beyond the bridge span. The first contour line that connects the left and right abutments can be used as a starting point for determining a WSE to be used for freeboard determination. As mentioned above, these approaches should only be considered as a starting point, as WSE and determination of freeboard vary site to site.

8.1.2. Scour Analysis

Scour is the removal of material as the result of hydraulic forces acting upon sediment. Wherever the rate of sediment deposition is less than the rate of transport, scour occurs. Practitioners have historically predicted scour using empirical and theoretical equations. These equations link the hydraulic conditions acting on the channel to the resulting scour. Each scour equation is founded upon a different set of empirical data or theoretical perspective. Please refer to HEC-18 for more information on scour equations, their assumptions, and limitations.

Scour occurs over time in three dimensions. Standard approaches to evaluating scour are based on flume experiments that are relatively uniform and generally represent average characteristics. Standard scour equations were also developed in the context of obtaining results from analytical methods or 1D models. However, hydraulic results from a 2D model provide a more accurate representation of the flow distribution on which to base the averaged values. More specifically, a 2D model computes a more accurate representation of the flow distribution between the main channel and overbank areas, which is important to understand when evaluating scour potential. Engineers should become familiar with both evaluating 2D model results and the scour equations when using 2D models to assess scour.

Evaluating Contraction and Local Scour. To evaluate contraction and abutment scour, an approach section is defined to represent the un-contracted flow and sediment transport characteristics of the upstream reach. This approach section should be located upstream of the influence of the bridge encroachment, and its width should represent the approximate width of the flow that is transporting sediment. It is standard practice to use either the width between the tops of the channel banks or the width between the toes of the channel bank slopes. However, with an estimate of critical velocity for the bed material, 2D model velocity results can also be used to assess the width of flow that is truly capable of transporting sediment.

The 2D models' velocity vectors can also be used to identify the location at which overbank flows begin to be redirected toward the main channel as a result of the bridge encroachment on the floodplain. (Figure 8.3). Scour analysis uses average conditions in the main channel and overbank depending on the scour condition, and the bank station needs to be defined to specify the portion of the flow transporting sediment.

The ability to map unit discharge across the entire flow field is advantageous (Figure 8.3). The unit discharge shows the engineer the distribution and intensity of flow throughout the active model domain. Analyzing a map of the unit discharge provides insight into the distribution of flow within, and upstream of, a bridge crossing that can be used to determine possible approach cross section locations.

The HEC-18 equation 6.1 provides a method for the engineer to calculate the minimum velocity and critical velocity necessary to transport sediment, given a flow depth and grain size. The equation below shows how this equation can be used to calculate the critical velocity index from 2D model results.

$$\text{Critical Velocity Index} = \frac{V}{11.17H^{1/6}D^{1/3}}$$

(8.1)

where:

V   =   flow velocity (ft/s)
H   =   depth of flow (ft)
D   =   median grain diameter (ft)

If the critical velocity index for a location is greater than one, the hydraulic conditions have the intensity required to mobilize sediment. If the critical velocity index is less than one at a given location, the hydraulic conditions lack the intensity required to mobilize sediment. A critical velocity index map (Figure 8.3) of the entire model domain can be generated from 2D model results that provide insight for an engineer to determine where sediment may be mobilized.
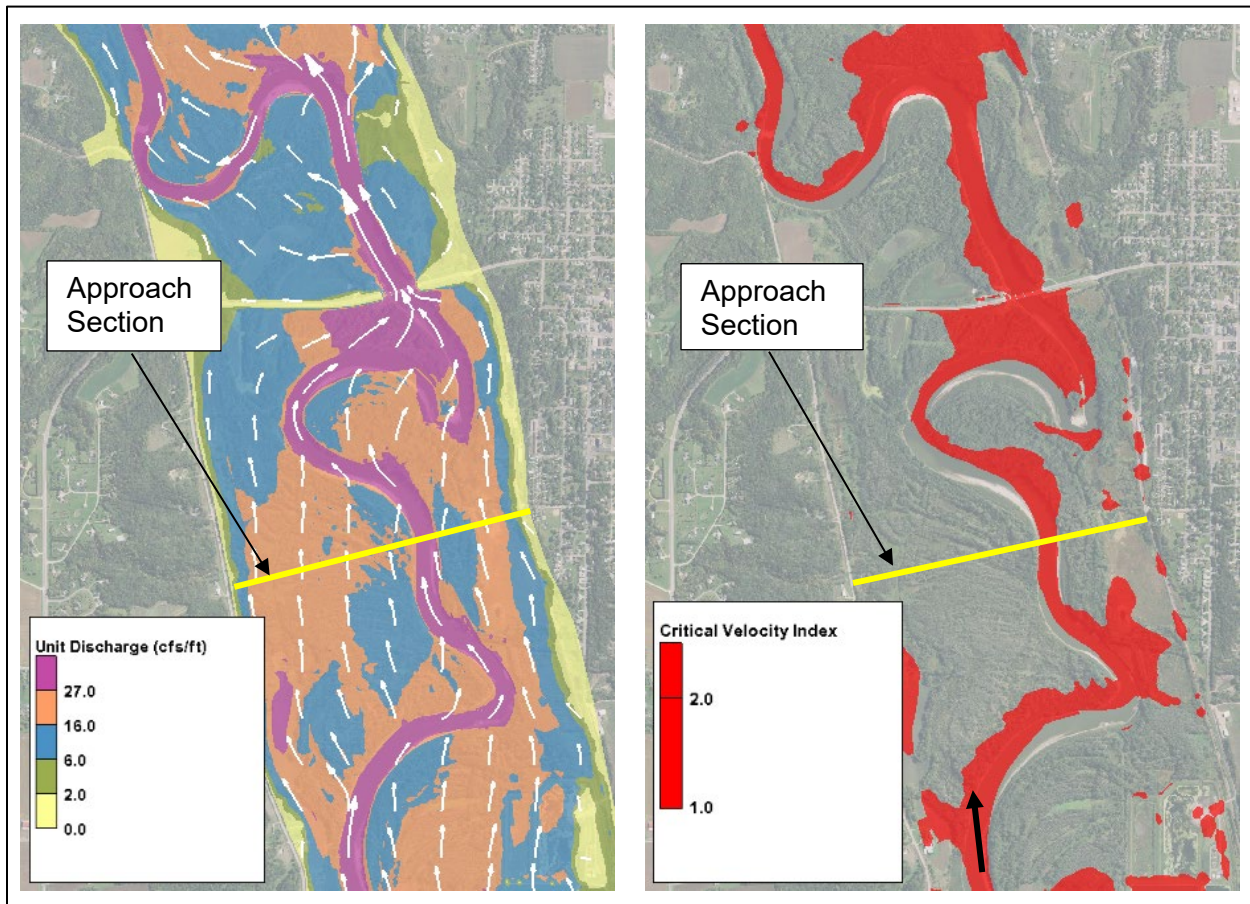
Figure 8.3. Approach cross section selection example; unit discharge map with velocity vectors (left), and critical velocity index map (right).

A critical velocity index value greater than one does not necessarily indicate scour at that location. Sediment transported from upstream may still deposit at that location at a rate equal to or greater than the rate at which sediment is being removed. The critical velocity index applies to well-sorted, sand-sized sediment with sizes above 0.2 mm.

The Shields parameter provides a good method for predicting the incipient motion of sediment particles by quantifying the critical shear stress for a specific size of sediment. Critical shear stress threshold results can be used to assist with a scour analysis (see Figure 7.6). Shields parameter values have been developed for a variety of sediment characteristics (i.e., sizes and gradations).

8.6

This allows using the Shields parameter for sediment conditions, under which the critical velocity index may not be applicable. The appropriate Shields parameter value should be selected with care, such that it applies to the sediment characteristics in the study area. It is therefore important to know the sediment characteristics for all representative locations (e.g., river channel, banks, and floodplains) within a study area, to select the appropriate Shields parameter value for each.

Evaluating Local Scour Hydraulics. While a 1D model can provide cross section averaged results and approximate velocity distribution at a cross section based on available conveyance, it cannot provide results at all points within the model domain. The ability to analyze hydraulic variables at specific points in a 2D model gives the modeler important insight into the factors driving scour.

Because 1D modeling methods cannot resolve flow concentration effects at bridge abutments, HEC-18 section 8.6.3 presents methods to adjust 1D modeling results based on the embankment and floodplain/channel configurations. A 2D model does not require the application of these often-excessively-conservative adjustments. Similarly, for pier scour computations, 2D models directly compute impinging velocity magnitude and direction, precluding the need to make conservative and often highly inaccurate impinging-angle assumptions.

Before extracting point-specific hydraulic results from a 2D model, it is necessary to understand mesh density as it relates to the accuracy of the results. For example, if large mesh elements are used to represent a bridge location with small piers, the model may not sufficiently represent the geometry of the pier or local hydraulics, and therefore may not generate a local flow field adequate for scour analysis. However, with smaller mesh elements, a 2D model can produce local flow field results for the area of interest providing an engineer with more precise depth, velocity, and flow direction (Figure 8.4.). This data can then inform the computations for pier scour, abutment scour, bend way scour, or even overtopping scour. The element spacing on the left image is too coarse and does not provide sufficient results. The right image provides slightly more detailed results than the middle image but uses more elements than necessary.
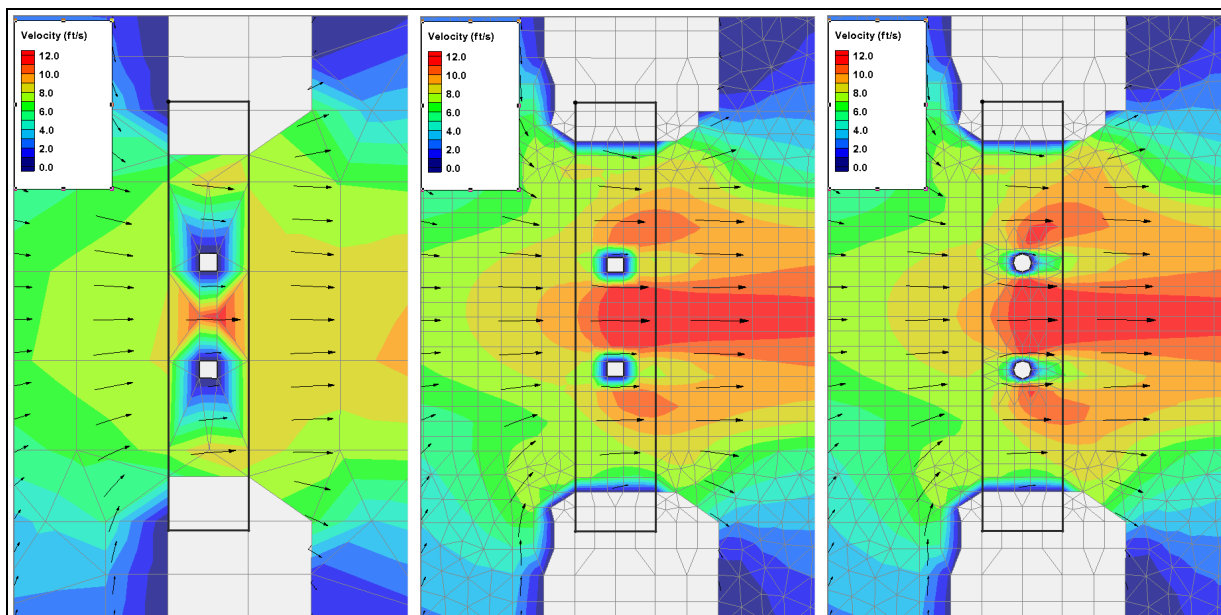
Figure 8.4. Velocity contours through a bridge with a coarse mesh (left), recommended mesh with simple pier (middle), and overly detailed pier (right).

8.1.3. Bank Stabilization

As noted in Chapter 3, 2D models determine hydraulic parameters at each computational element or node, while a 1D model only provides averaged hydraulics at cross section locations specified by the modeler. The spatial representation of 2D results is useful in determining the magnitude and location of maximum velocity, or the bed shear stress exerted by the flow at a specific location (e.g., the location of bank stabilization). Conversely, 1D models cannot determine the maximum velocity or bed shear stress exerted on a scour countermeasure, which can introduce a potentially significant error in the design or result in overly conservative measures.

Recognizing this limitation, empirical functions that predict the maximum value of a hydraulic variable based on its average value have been developed. For example, determination of the riprap size for a scour countermeasure is based on a design flow velocity applied to the riprap at a given cross section (HEC-18). Because 1D models only calculate the average velocity, an empirical relation for estimating the design velocity from the cross-sectionally averaged flow velocity is required. Although this empirical relation enables the use of 1D models for designing riprap bank protection, it introduces additional uncertainty into the design. Results from 2D models can be used to determine riprap sizing that provides a more detailed assessment of bank stabilization requirements (Figure 8.5).
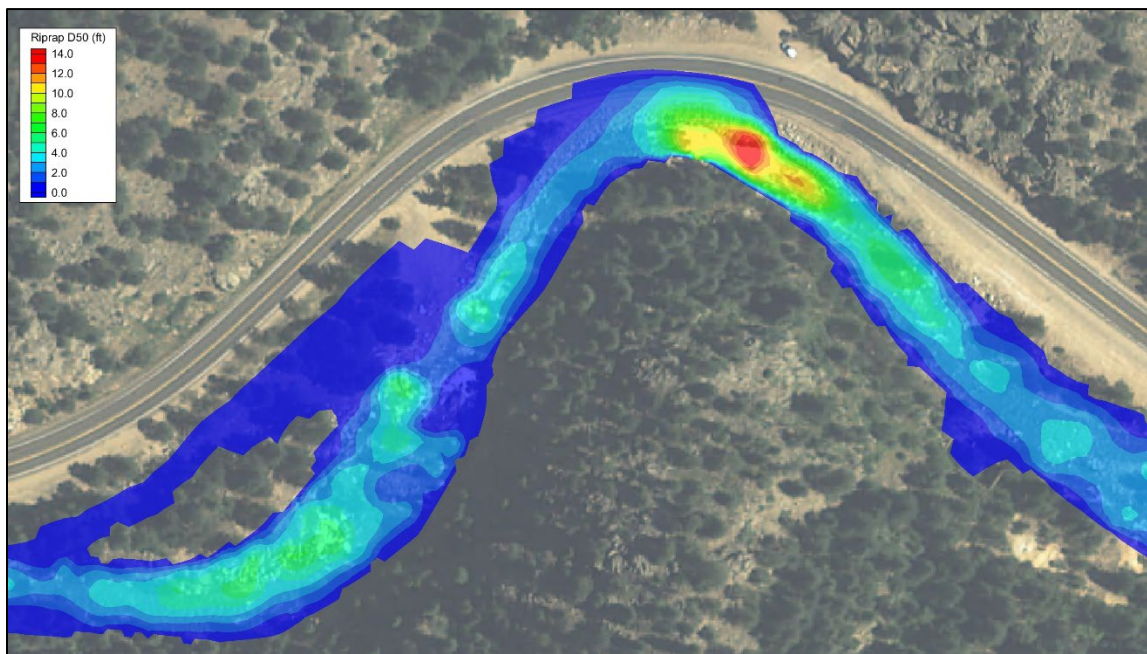
Figure 8.5. Riprap sizing contours based on 2D model depth and velocity results.

Because 2D models provide more accurate results, they are advantageous when designing scour countermeasures that impact the flow path, such as barbs, dikes, and bendway weirs. These in-stream, flow deflection structures introduce localized contraction and expansion to the flow and lead to flow separation. Two-dimensional models explicitly resolve these local flow patterns as can be observed in the computed hydraulics (Figure 7.4). Additionally, more accurate and detailed 2D model results at scour countermeasures are beneficial for selecting the spacing between a series of such structures. In contrast, 1D models cannot inherently capture the effects of lateral flows and flow separation, thus requiring the use of empirical coefficients, which can compromise the reliability of the hydraulic results.

8.2. FLOODPLAIN IMPACT ANALYSIS

Accurate hydraulic information is necessary to assess impacts of activities within floodplains (regulatory and non-regulatory). Two-dimensional models are excellent tools that provide decision-makers with data required for use in floodplain analysis. Common floodplain analysis includes assessing hydraulic impacts of proposed activity within the floodplain, determining various areas of risk and regulatory floodplain zones such as floodways, and designing solutions to mitigate damages from future flood events.

River hydraulics are highly dynamic during significant flood events and often include variable discharges, active erosion and deposition, bank failures, debris, and blocked culverts or bridges, all of which are difficult to predict and model. While 2D models are excellent tools to aid in floodplain analysis, they still have limitations when attempting to analyze real-world flood events.

According to FEMA, floodplain management is "…a decision-making process that aims to achieve the wise use of the nation's floodplains. Wise use means both reduced flood losses and protection of the natural resources and functions of floodplains." See https://www.fema.gov/floodplain-management-definition. While transportation engineers do not have the official role of floodplain manager, working within floodplains does involve the understanding and management of water during flood events and associated impacts on the built and natural environment. Therefore, anyone working on projects in floodplains should understand floodplain analysis, management, and mitigation.

8.2.1. Assessing Impacts

Most hydraulic modeling studies, and especially floodplain management analysis, involve comparing hydraulic results between multiple scenarios. A typical situation involves comparing existing and proposed conditions. Other comparisons might involve historical conditions versus existing conditions or a sensitivity analysis between the same model geometry configuration with different model settings such as roughness type or time step. A situation applicable to this document is a bridge replacement project with adjacent approach roadway work and the potential hydraulic impacts to the bridge, roadway, and adjacent property and structures.

Floodplain impact analysis can be completed using a 1D model, a 1D model informed by a 2D model, or a 2D model. Current floodplain regulations have been written in the context of using 1D models to develop floodplain maps, determine floodways, and assess impacts of proposed changes within floodplain boundaries. However, 2D models provide more realistic insights into differences of a variety of analyzed conditions instead of assumed, averaged, and interpolated results courtesy of 1D models. This is why 2D models have been used to inform 1D models that become the regulatory model.

For example, a 2D model depicts the spatial variation in WSE when comparing two different model scenarios of the same location. In the real world, any geometric change within a floodplain produces a local alteration in flow patterns. Consequently, any obstruction in the floodplain, even if offset by adjacent grading cuts or other increases in floodplain flow capacity, typically causes a local WSE increase, and corresponding decrease, in WSE either upstream or downstream of the change in floodplain configuration.

Considering the likely impacts on WSE, to meet floodplain regulations using 2D models, practitioners should engage in discussions with local floodplain regulators and potentially local FEMA representatives. Approaches to meeting FEMA requirements may vary, but meeting "no-rise" criteria is still possible. See, e.g., 44 C.F.R. 60.3(d)(3) (providing a no-rise requirement for

certain encroachments). However, it is important to determine the specific criteria and procedures to be used to assess a no-rise. Examples of potential methods might include using an average WSE across the floodplain, determining some distance upstream of the project to measure impacts, assessing that no adverse impacts have occurred, or other approaches determined appropriate. Some states have state-specific no-impact requirements.

The typical hydraulic variable assessed for floodplain management projects, and the variable FEMA requirements focus on, is the WSE. While this is important, it is not the only hydraulic impact projects might have during a flood event. Project impacts to velocity, depth, shear stress, and the redistribution of flow throughout the floodplain have the potential to impact transportation infrastructure, surrounding properties, and natural resources. While FEMA does not require analysis of these impacts, it is recommended they are reviewed to determine actual hydraulic impacts of the bridge or project feature.

As an example of the more complete data a 2D model provides, a bridge replacement and roadway project may involve an increase in hydraulic capacity of the bridge opening while raising the adjacent roadway to increase the discharge at which overtopping occurs. For bridge hydraulics, the WSE can vary significantly along a cross section upstream and downstream of the bridge. A 1D model study would provide one WSE value at each cross section adjacent to the roadway. Example plots of existing WSE and difference in water surface due to proposed conditions are shown in Figure 8.6 and Figure 8.7. A profile through the proposed bridge and a cross section along the roadway showing both WSE's is shown in Figure 8.8 and Figure 8.9.

The existing condition WSE and cross section plots show the overtopping of the roadway embankment. The WSE overtopping the roadway is about 1.5 feet higher than the corresponding WSE in the channel as seen in the cross section. The wider proposed bridge results in a localized decrease in the WSE upstream of the bridge location. However, due to the increased flow allowed into the channel from the larger hydraulic opening, there is also a localized increase in WSE downstream of the proposed bridge. A 1D model would typically not show this downstream WSE increase due to how 1D model redistributes flow within the downstream cross section. The apparent WSE increase locations on the upstream side are due to changes in grading near those locations and do not represent actual increases.
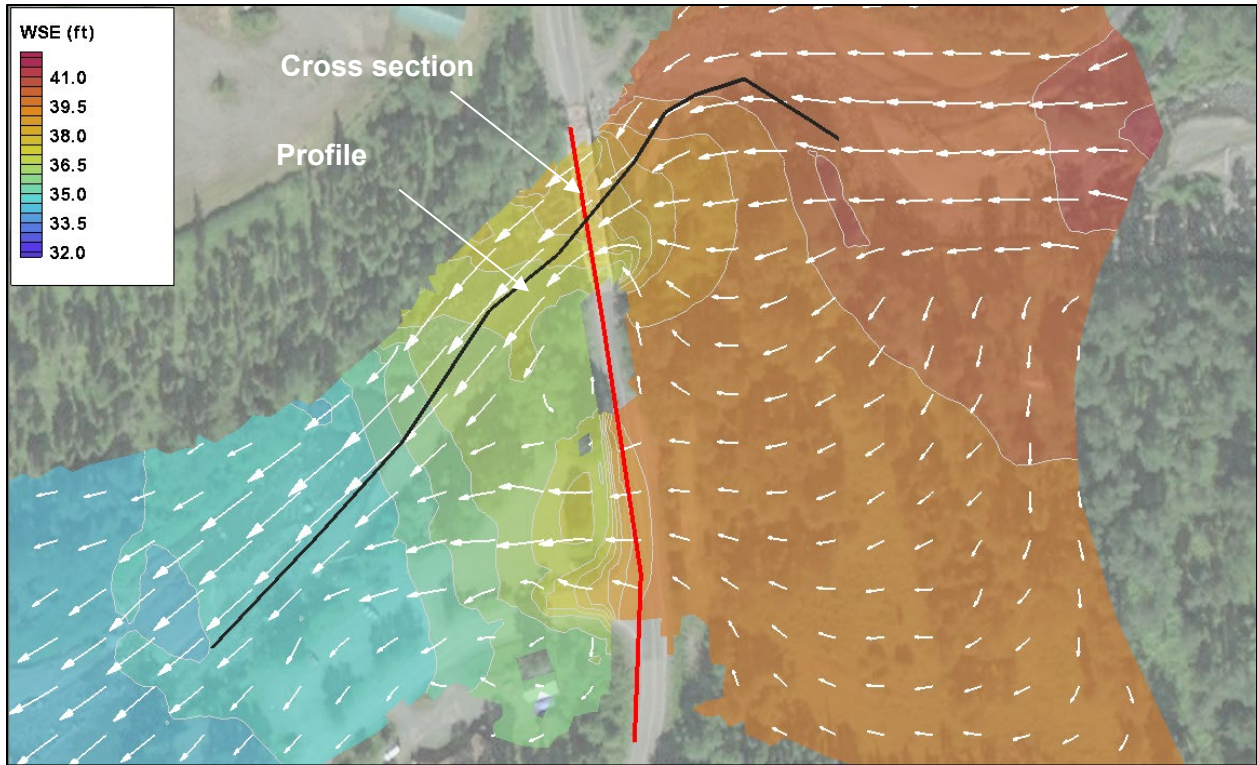
Figure 8.6. Plan view of existing conditions WSE at a bridge crossing.
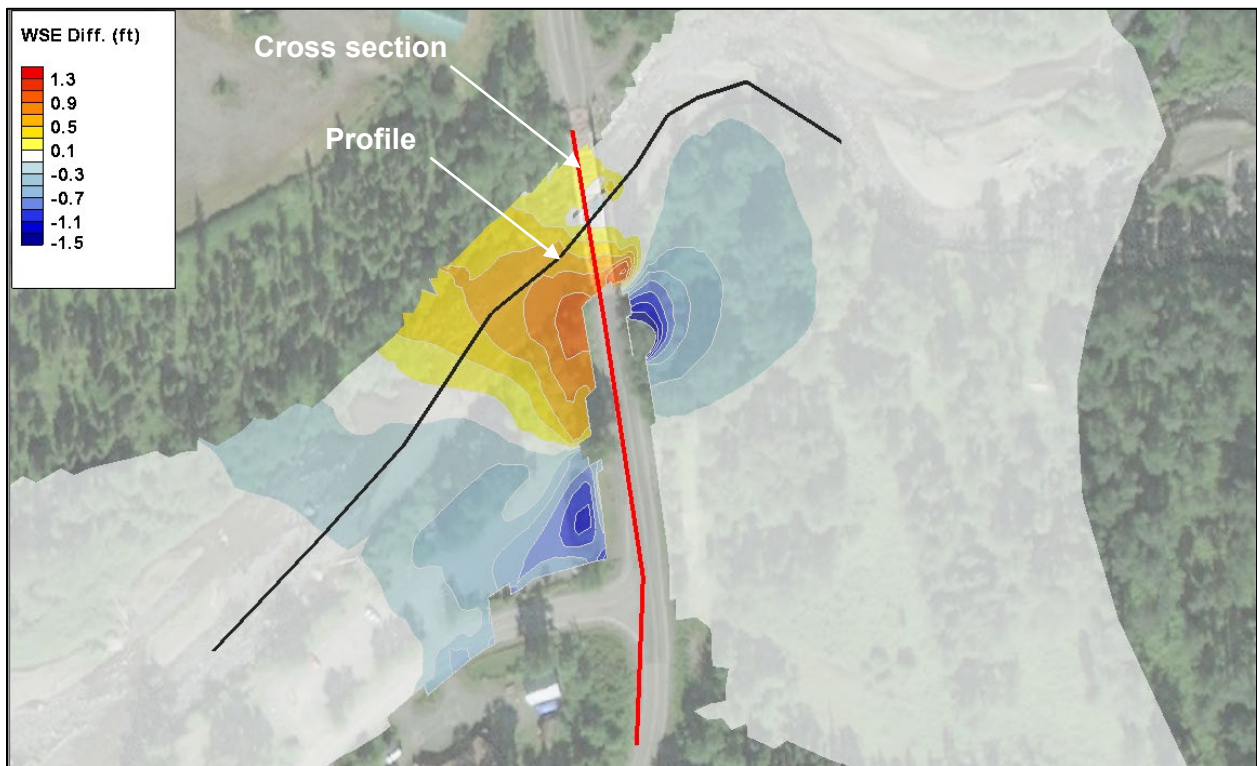
Figure 8.7. Plan view of change in WSE, proposed vs. existing condition.

8.11
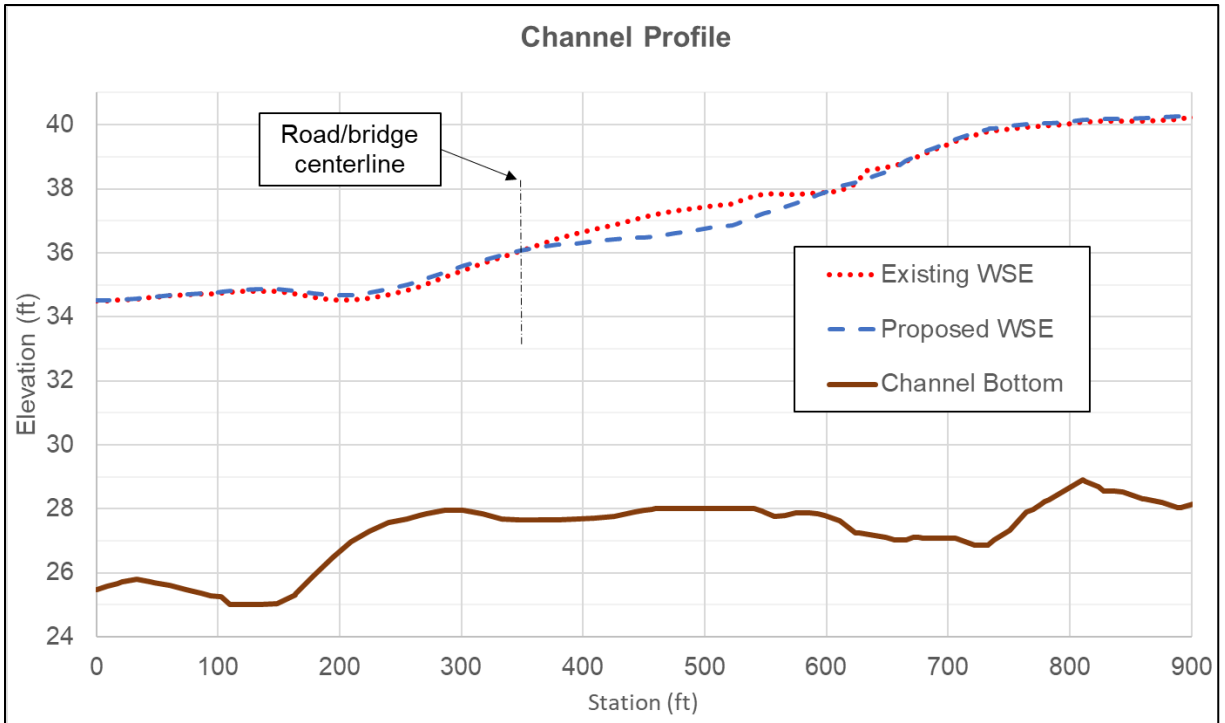
Figure 8.8. Profile plot of channel and WSE of existing embankment and proposed condition along channel centerline.
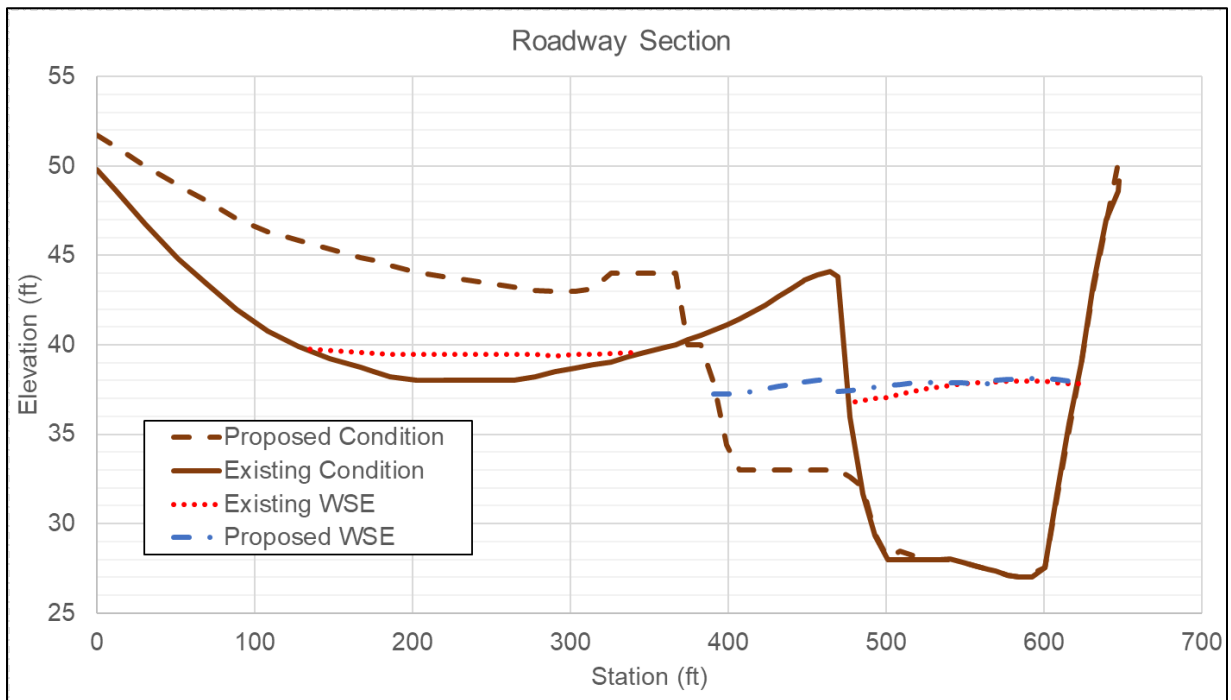
Figure 8.9. Cross section of proposed and existing ground and WSE through road centerline.

8.12

8.2.2. Floodway Concepts

The regulatory floodway is an important element of the National Flood Insurance Program (NFIP). The regulatory floodway means "the channel of a river or other watercourse and the adjacent land areas that must be reserved in order to discharge the base flood without cumulatively increasing the water surface elevation more than a designated height." 44 C.F.R. § 59.1. To determine the "designated height," certain communities required by regulation to adopt such floodways must ensure that "the area chosen … [can] carry the waters of the base flood, without increasing the [WSE] of that flood more than one foot at any point." 44 C.F.R. § 60.3(d)(2). In a given NFIP floodplain, the regulatory floodway is the area between the floodway encroachment lines, often depicted as a hatched area within the 100-year floodplain on a FEMA Flood Insurance Rate Map (FIRM). See 44 C.F.R. § 59.1 (defining "floodway encroachment lines"). Figure 8.10 provides a cross section view of a floodplain illustrating the floodway concept. Figure 8.11 is an excerpt of a FIRM that includes a regulatory floodway. The floodway is considered critical for flood flow conveyance. Encroachments such as fill, new construction, or other development in the regulatory floodway trigger no-rise performance standards; i.e., these encroachments must "not result in any increase in flood levels within the community during the occurrence of the base flood discharge." 44 CFR 60.3(d)(3).
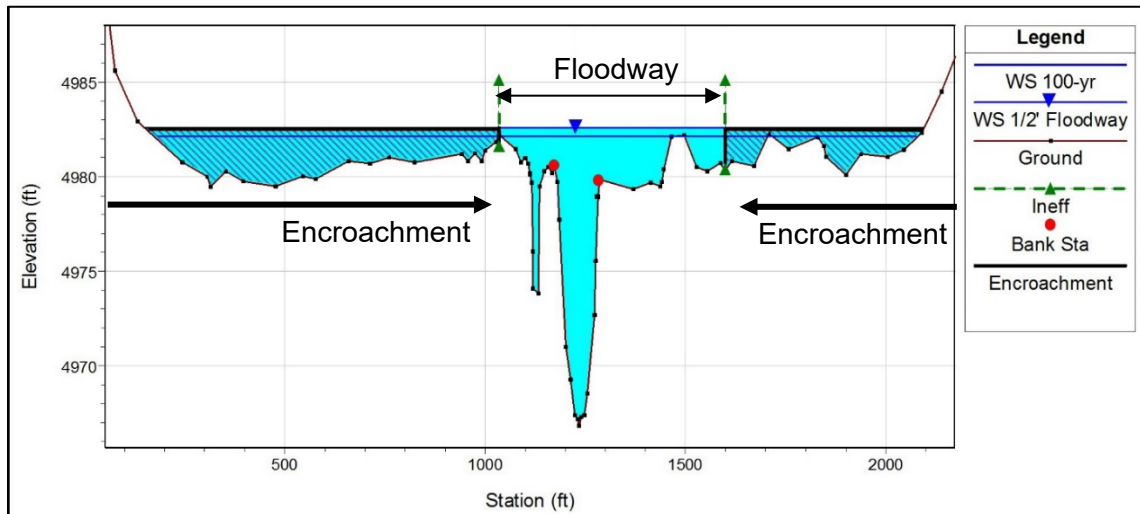
Figure 8.10. Typical cross section showing 1D model ½-foot floodway encroachments.

8.13
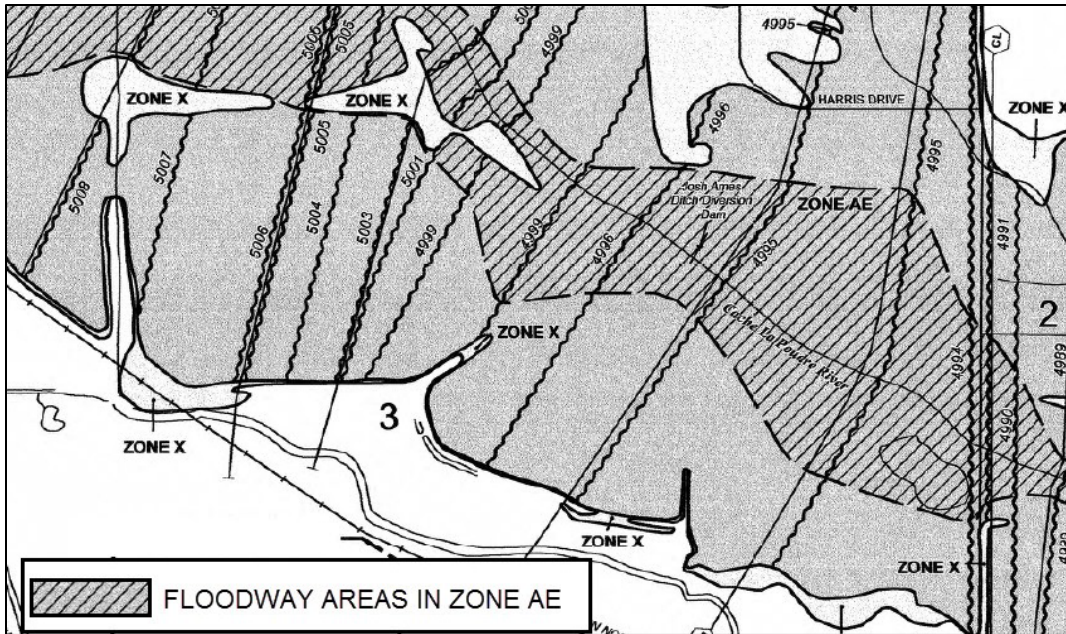
Figure 8.11. Typical FIRM illustration of the floodway.

If the FIRM for a floodplain depicts a regulatory floodway, the community having jurisdiction over that floodplain is required to, as a condition of remaining in good standing in the NFIP, prevent encroachment within the floodway unless it is demonstrated that the encroachment will not result in any rise in the 100-year flood profile (44 CFR 60.3(d)(3)). Enforcement of this requirement is generally incorporated into the local floodplain development permit process. Id. (requiring communities to prohibit such encroachments). If a state transportation agency is the project proponent, the floodplain development permit is sought from the local jurisdiction (e.g., city or county), or a state agency having NFIP authority.

FEMA's floodway guidelines call for setting the encroachment limits so that properties on both the left and right side of the river are generally treated equitably. For decades, floodway analysis in the NFIP has been performed primarily using 1D modeling. In the 1D context, encroachment limits are typically considered equitable if the reduction in conveyance caused by the encroachment is the same on both the left and right overbanks. One-dimensional modeling programs such as the USACE program HEC-RAS include automated tools for finding approximate floodway encroachment limits at every cross section that satisfy the objective of equal conveyance reduction on the left and right overbanks.

Transferring the floodway concept to 2D modeling has thus far proven challenging. The equal conveyance concept is very amenable to 1D modeling because all computation nodes are cross sections. Equal conveyance reduction is not readily applied in 2D modeling. Consequently, alternative methods are needed to satisfy the equitable treatment requirement if using a 2D analysis for floodways. A promising method uses contours of equal unit discharge as trial encroachment lines on both sides of the main channel along the length of the floodplain in the model domain. This approach acknowledges that the floodway should encompass a dominant portion of the active flow width in a floodplain.

Allowable surcharge is another aspect of the floodway concept difficult to transfer from the 1D to 2D modeling context. In the 1D context, under which the floodway idea was developed, the surcharge is evaluated at every model cross section, comparing the WSE for encroached versus

unencroached conditions. In this comparison, both the encroached and unencroached WSE are considered constant across the entire active width of the cross section and are based on a single conveyance, friction slope, and velocity head for the cross section. As demonstrated throughout this reference document, the actual WSE does vary, sometimes significantly, along the width of a cross section. A 2D model calculates this variation.

When developing the floodway in a 2D model, the surcharge limitation and calculated surcharge could be defined on the basis of maximum surcharge at any node, or by plotting contours of water surface difference between the encroached and unencroached simulations. Using the latter approach with the current definition and evaluation of surcharge will likely result in wider floodways from 2D models compared to 1D models. The inherent averaging of 1D models obscures the reality that the actual surcharge at one or more locations along a cross section is greater than the value represented by comparing two average water surface values. Preliminary analysis of floodway limits developed from current standard of practice applied to 2D models shows significantly more (two feet or greater) WSE rise than the assumed one foot of allowable rise.

Any discussion of the issue of defining and evaluating floodway surcharge reveals the need for a thorough re-examination of the floodway concept, its purposes and benefits, and how to honor those purposes and preserve those benefits in the future as 2D modeling becomes the norm rather than the exception. Further research is being conducted to provide resources to DOT agencies on how to meet federal or local floodway regulations when using 2D models.

8.2.3. Floodplain Mitigation

Flood mitigation involves managing floodwater to reduce the loss of life and property due to flooding. Removal of infrastructure from the floodplain is often considered the best flood mitigation solution. However, it is typically not feasible to remove all infrastructure from floodplains at all levels of flooding. Floodplain mitigation designers work with floodplains and sometimes river channels to create solutions that reduce risks to life and minimize impacts to or rehabilitate the natural function of riparian corridors

Understanding the risk from flooding is a vital part of effective mitigation and requires access to hydraulic conditions during a variety of flooding discharges. Two-dimensional models can provide an accurate understanding of local flood conditions and help guide appropriate flood mitigation solutions by answering important questions such as:

- What is the source of flooding?
- How much flow is there in a specific location?
- What is the depth or velocity at a specific location?
- What is the direction of flow?
- What are the primary flow paths of floodwaters?
- How much water flows through a bridge, culvert?
- How much water flows over a roadway?

The current FEMA floodplain modeling and mapping processes were created to help develop approximate flood insurance rate maps for the NFIP. FEMA maps are insurance rate maps, not detailed flood risk maps or tools to be used to assess appropriate mitigation solutions. They are also not typically at a scale for sufficient hydraulic analysis of transportation infrastructure such as bridges and culverts.

For locations with complex hydraulic conditions, existing maps and models are typically not adequate to fully understand, or assist in appropriately mitigating against, many of the nation's flood problems. Though it is standard practice to use these tools to attempt to solve such problems, the distribution and movement of flow within the floodplain, as well as the interaction of flow between the channel and floodplain, is of primary importance when considering various flood mitigation solutions. Having a more complete understanding of the hydraulics related to a project may not be a FEMA requirement, but is invaluable to engineers when working with transportation infrastructure in the river environment.

One-dimensional models cannot sufficiently provide this type of information for complex hydraulic locations. One of the primary assumptions of a 1D model is that the flow is primarily "one-dimensional," i.e., in the downstream direction. This assumption is violated for medium to wide floodplains and when bridges and approach roadways cross floodplains.

A standard 2D model can be used to assess how flood events impact a project and what impacts a project has on the floodplain and adjacent properties. Typical flood mitigation includes the steps outlined in Figure 8.9.
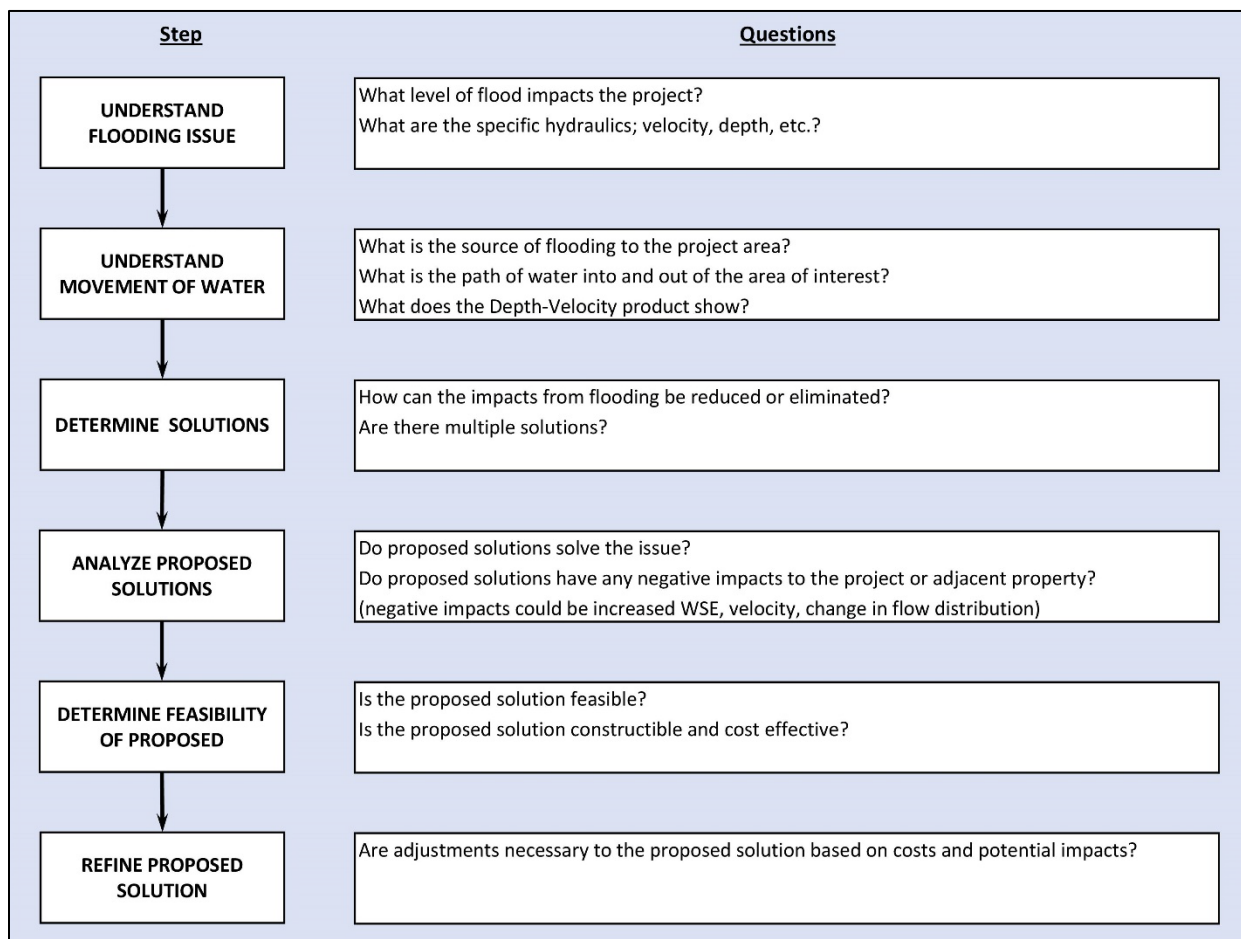
Figure 8.12. Flood mitigation standard workflow.

8.3. RIVER REHABILITATION AND AQUATIC HABITAT ASSESSMENTS

Stream or river rehabilitation is an ever-growing field and applies to the design of infrastructure in riverine settings. Two-dimensional models provide rehabilitation practitioners with more accurate data for the design of transportation infrastructure that minimizes effects on the surrounding environment. Two-dimensional model results can provide superior graphical visualizations that aid in communication with resource agencies and stakeholders to show the benefits and potential effects from various project alternatives.

Figure 8.13 and Figure 8.14 show existing conditions in a stream reach that has been modified over the last 100 years. As seen in the figures, some portions of the floodplain are not activated at high flows (e.g., 10-year flow), and thus additional energy is exerted on the channel bed and banks, which has caused the system to incise over time. These figures highlight the areas with higher velocities and shear stresses that cause incision, in addition to areas where continued bank erosion could be expected.

A 2D model can be used by the rehabilitation practitioner to develop a design that mimics natural stream processes, such as activating floodplains, re-establishing side channels, and creating diversity in flow characteristics for aquatic habitat. These types of natural stream processes are not an output from a 1D model and need the modeler to determine potential flow paths rather than relying on the site's terrain. In addition to the areas with no floodplain interaction, Figure 8.13 and Figure 8.14 also show the diversity in flow characteristics during a 10-year flow event, including where side channels are activated which may not be accurately determined with a 1D model.
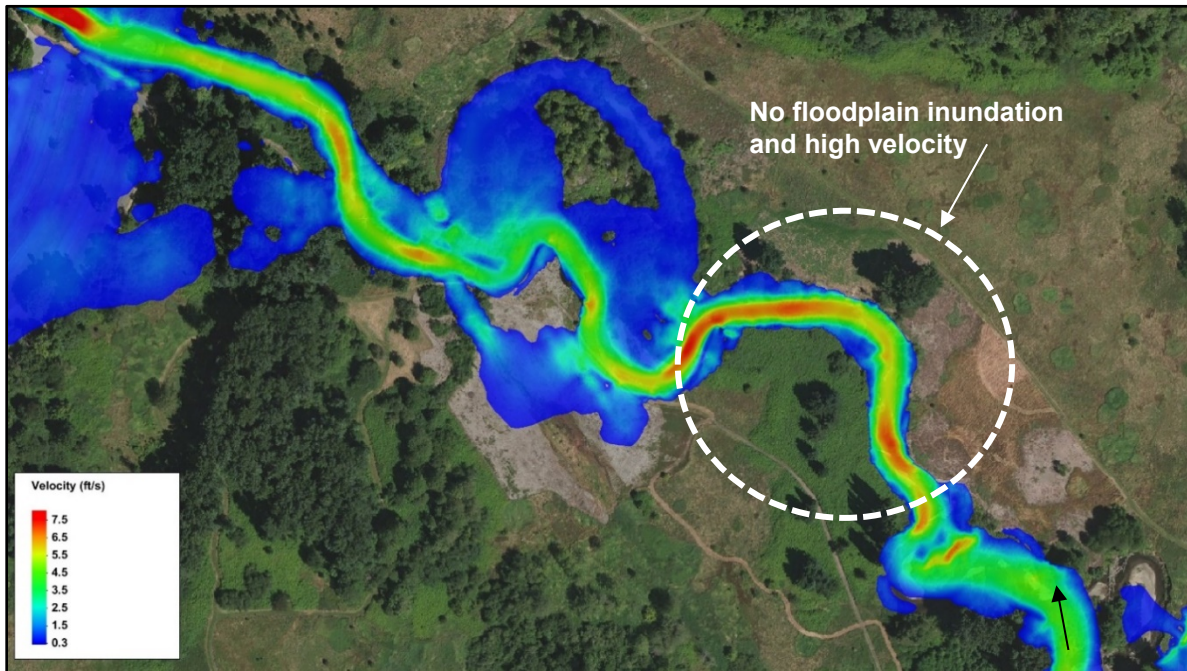
Figure 8.13. Plan view of velocity contours calculated by a 2D model for a stream rehabilitation project.
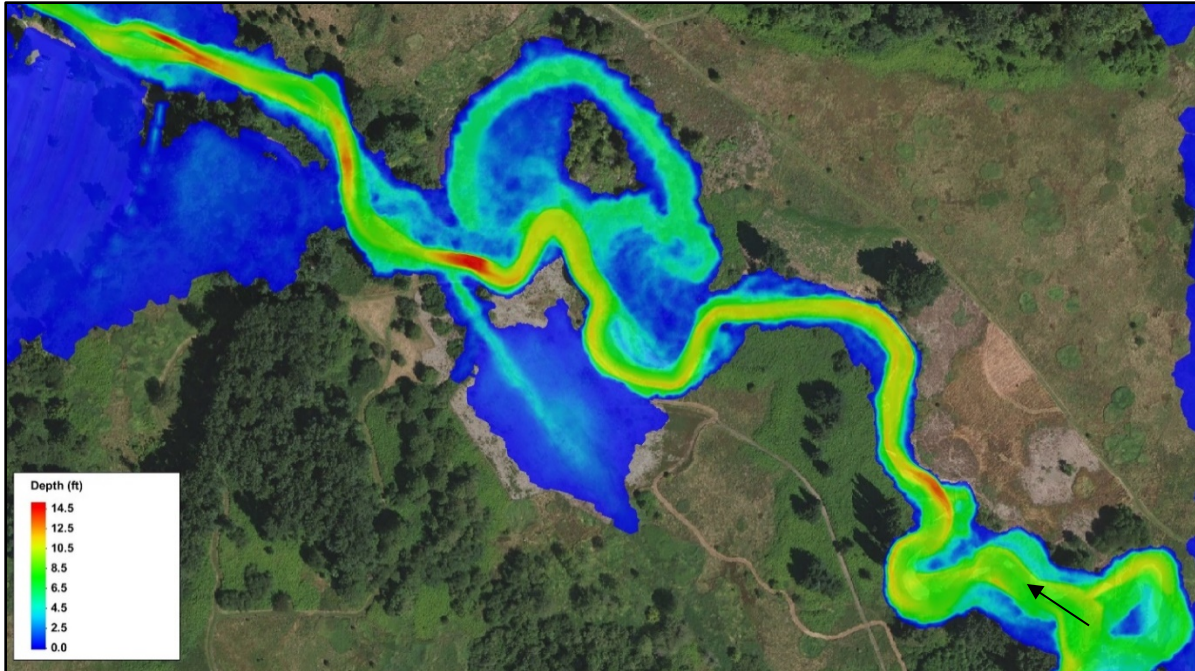
Figure 8.14. Plan view of flow depth contours calculated by a 2D model for a stream rehabilitation project.

Two-dimensional model results, such as flow depth, velocity, and shear stress, can provide a better understanding of the diverse nature of flow characteristics (compared to 1D model results) that are typically correlated with aquatic habitat in a variety of aquatic habitat analyses.

Various hydraulic structures can be used to create habitat and rehabilitate/restore natural river/stream processes such as barbs, groins, large woody material (LWM), and engineered log jams (ELJs). Understanding the diversity of hydraulic characteristics for these types of structures can be important to both the hydraulic design as well as the biologic effects on aquatic habitat.

As an example, Figure 8.15 depicts a project which was designed to protect a state highway while providing habitat to aquatic species. The project included a series of ELJs constructed along the highway prism and an apex ELJ located on a mid-channel gravel bar. The 2D model mesh was refined in the areas of the ELJs to properly reflect their geometry and obtain hydraulic characteristics at a scale, which would support their design. In addition to being used for design, the 2D model results (bed shear stress) in Figure 8.15 were overlain with surveyed locations of Chinook salmon redds (yellow dots). Based on this flow condition, it appears the Chinook redds locations may correlate well with a shear stress around 1 to 2 lb/ft^2. This type of assessment can provide an understanding of suitable flow characteristics that a designer may want to replicate to improve aquatic habitat or determine potential effects a project may have on existing habitat.

8.18
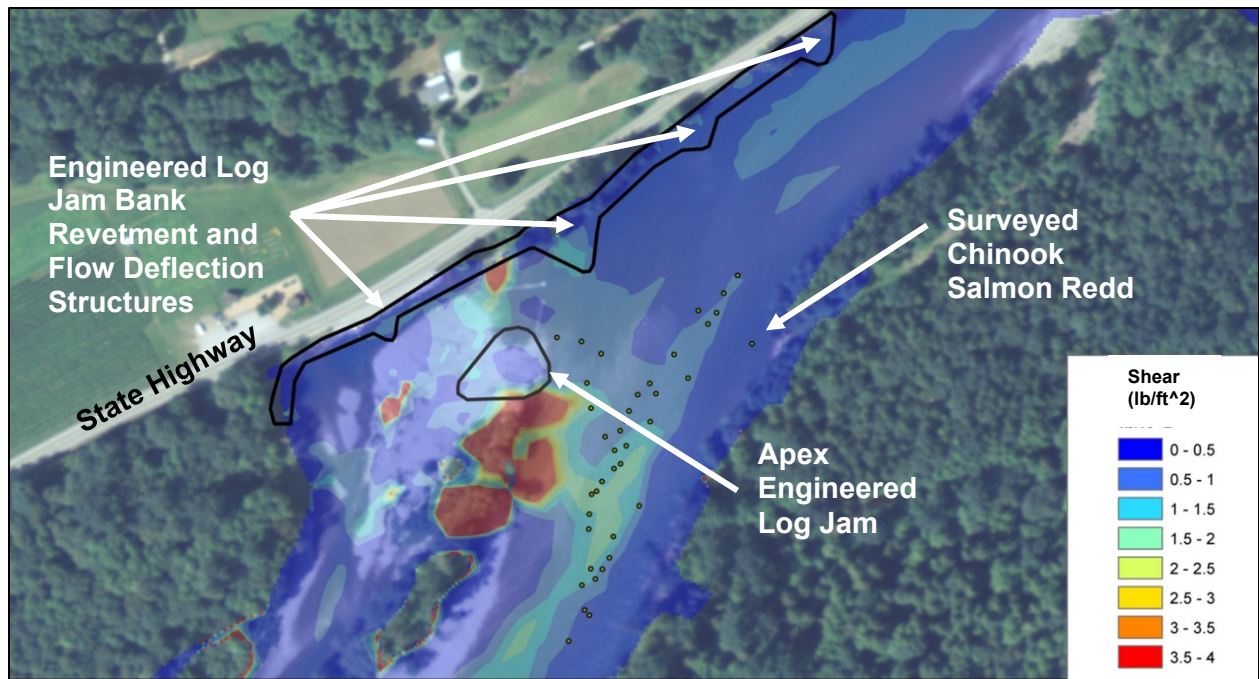
Figure 8.15. Surveyed chinook salmon redds with 2D model bed shear stress contours.

An additional application of 2D model results is to determine if an anthropogenic or natural barrier to fish migration exists within a project reach (Figure 8.16). Natural streams and rivers often have complex hydraulic diversity, which can be beneficial to aquatic habitat. One-dimensional models do not provide or illustrate the diversity in hydraulic characteristics that may assist with showing areas of suitable habitat or conversely determining if a fish barrier exists. Figure 8.16 shows an example of a natural stream reach (top photo) with 2D model outputs of flow depth (middle photo) and flow velocity (bottom photo). These model results were assessed by both hydraulic engineers and fisheries biologists to determine if a fish barrier existed, and assisted in the development of alternative designs to potentially remove any identified barriers.
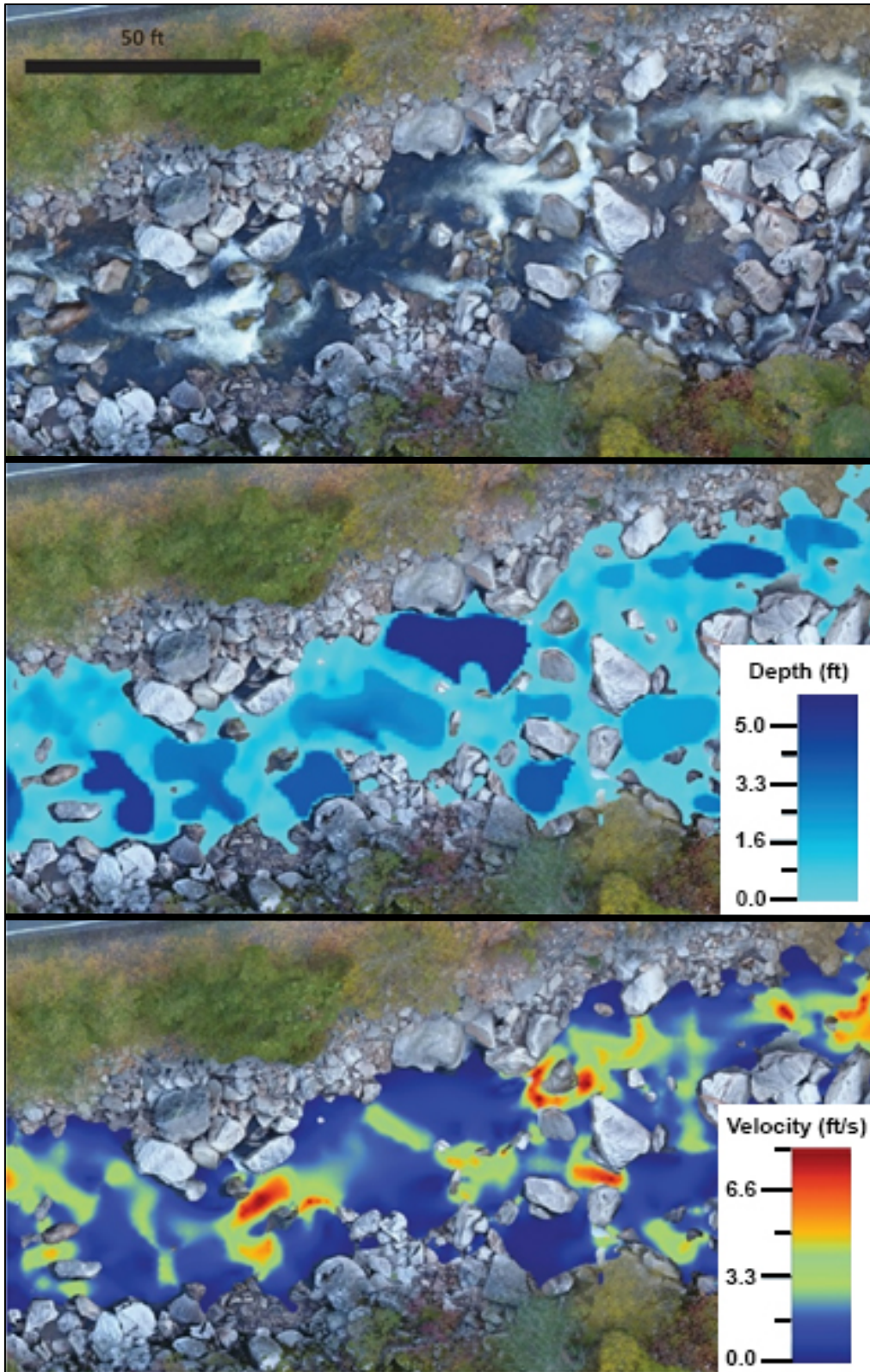
Figure 8.16. Fish passage barrier assessment - depth (middle) and velocity (bottom).

8.4. SEDIMENT TRANSPORT

River systems are inherently dynamic. Flowing water is capable of mobilizing and transporting materials in the water column, including sediment. The interaction between flow and the terrain over time defines both the channel and the floodplain and significantly alters the riverine system. Modeling this complex interaction is a challenging data collection and numerical analysis problem. However, modeling how a river transports sediment and the changes flowing water makes to the terrain can provide significant design insight. Mobile-bed sediment transport hydraulic models predict the changes in the channel and inundated overbank as flowing water erodes and deposits sediment. These models present additional complexities compared to standard 2D models, and are not applied to projects as widely due to the following reasons:

- Sediment transport models are numerically intensive.
- Extensive data is required for channel and overbank sediment characteristics.
- Additional hydrologic analysis is required to develop representative dynamic hydrographs.
- Geotechnical stability is not currently accounted for.
- Significant inherent uncertainty exists in the equations used to evaluate sediment transport.
- Calibration and verification of sediment transport data sets are required if used directly for quantitative analysis of sediment transport.

However, mobile-bed models are beneficial tools and may be necessary to accurately and quantitatively evaluate erosion and deposition patterns for some design problems. Examples of the kinds of design problems suitable for 2D mobile-bed hydraulic analysis include:

- Dam breach/removal: evaluating how the former reservoir and channel system responds to the loss of base-level control and the altered sediment transport regime as the dam fails or is removed.
- Contraction scour analysis
- Dam or in-channel sediment mine installation

They may also be used to gain qualitative insight into sediment transport problems that have significant 3D hydraulic components, such as:

- Pier scour
- Abutment scour
- Bendway weirs
- Spur dikes
- Long-wall scour

Ongoing research on the use of 2D models for developing empirically informed relationships to account for 3D hydraulic effects points to the potential future application of mobile-bed technology on transportation-related problems like these.

Many 2D models include various sediment transport capabilities, which use sediment transport equations to evaluate the amount of material a given flow can carry. It is beyond the scope of this document to direct the selection of suitable sediment transport equations, but the user should select an equation that meets the following criteria:

- Suitable for the bed materials present in the system.
- Developed for the range of hydraulic conditions expected for the design region.
- Ability to be calibrated to provide reasonable results using available sediment-transport data for the design reach.

Some 2D sediment transport models can also adjust the underlying topography as the model predicts erosion or deposition based on predicted hydraulic conditions. In Figure 8.17, red and blue shades indicate areas of the modeled river reach around a bridge, where the 2D sediment transport model predicted erosion and deposition, respectively. These models are referred to as mobile-bed models. Such models perform up to five computational processes at each element for each time-step including:

1. Calculate 2D flow field as described in Chapter 2 and Chapter 3.
2. Calculate the sediment transport capacity using a suitable set of sediment transport equations, adjusting for cohesive effects if applicable.
3. Calculate the change in terrain elevation.
4. Calculate the change in in-situ sediment characteristics (gradation, porosity, consolidation, etc., if applicable).
5. Adjust bed for geotechnical stability (if considered).

Most mobile bed models currently do not directly consider geotechnical soil stability. However, soil stability considerations play a significant role in channel migration and ideally should be considered for direct analysis of stability trends in fluvial systems.

Multiple resources, including HDS-6 and "Erosion and Sedimentation" (Julien 2008), provide information for selecting an appropriate sediment transport equation. Because sediment transport is inherently uncertain and even best-fit sediment transport equations may result in absolute errors, sensitivity analysis, and calibration to robust field data sets is a necessary component of a mobile-bed analysis. Sediment transport models are therefore often used in comparative analysis and pattern definition versus predicting absolute values of erosion and sedimentation.
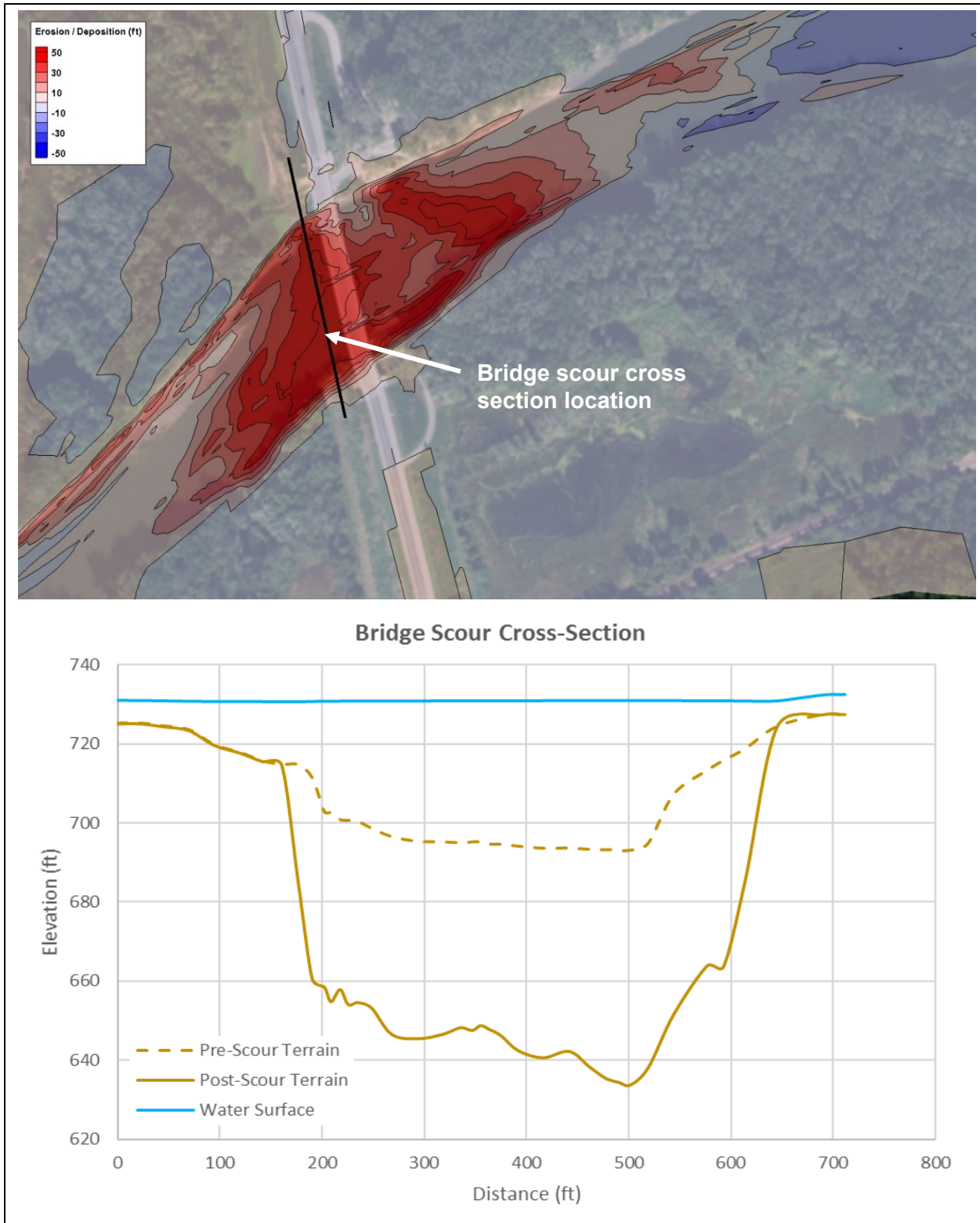
Figure 8.17. Example of mobile-bed 2D model results showing erosion and deposition due to contraction scour in plan and cross section views at a bridge crossing.

CHAPTER 9

REFERENCES

9.1. LITERATURE CITED

Alemseged, T.H., and Rienties, T.H.M, 2007: Uncertainty Issues in Hydrodynamic Flood Modeling. Proceedings of Spatial Data Quality 2007, 5th International Symposium, 13-15 June 2007, ITC, Enschede, The Netherlands.

American Society for Photogrammetry and Remote Sensing (ASPRS), 2014, "ASPRS Positional Accuracy Standards for Digital Geospatial Data, Edition 1, Version 1.0.)

American Society for Photogrammetry and Remote Sensing (ASPRS), 2018, "Digital Elevation Model Technologies and Applications: The DEM Users Manual, 3rd Edition," ASPRS, Bethesda, Maryland (Maune, D.F. and Nayegandhi, A.).

American Railway Engineering Association (AREMA), 2011, "Bulletin of the American Railway Engineering Association, Volume 13, No. 141".

Bagnold, R.A., (1980). An empirical correlation of bedload transport rates in flumes and natural rivers. Proc. Roy. Soc. Lond. Ser. A 372, 453-473.

Chow, V.T., 1959, "Open-Channel Flow," Second Edition, Springer Science and Business Media, New York, NY.

Cowan, W.L., 1956, "Estimating hydraulic roughness coefficients: Agricultural Engineering," v 37, no. 7.

Courant, R.; Friedrichs,K.; Lewy,H., 1928, "Über die partiellen Differenzengleichungen der mathematischen Physik", Mathematische Annalen (in German), 100 (1): 32–74, Bibcode:1928MatAn.100...32C, doi:10.1007/BF01448839, JFM 54.0486.01, MR 1512478.

Diplas, P., et al, 2008, "Sediment Transport Measurements," USGS Chapter 5 (Kuhnle, R., Gray, J., Glysson, D., Edwards, T.).

Federal Emergency Management Agency (FEMA), 2014, "Guidance for Flood Risk Analysis and Mapping, Vertical Datum Conversion," Guidance Document 24.

Federal Highway Administration (FHWA), 1978, "Hydraulics of Bridge Waterways," HDS-1, Second Edition, U.S. Department of Transportation, Washington, D.C. (Bradley, J.N.)

Federal Highway Administration (FHWA) 1986, "Bridge Waterways Analysis Model: Research Report, "FHWA/RD-86108, July.

Federal Highway Administration (FHWA), 2001, " River Engineering For Highway Encroachments – Highways in the River Environment," FHWA Hydraulic Design Series Number 6, Federal Highway Administration, Washington, D.C. (Richardson, E.V., D.B. Simons, P.F. Lagasse).

Federal Highway Administration (FHWA), 2002, "Highway Hydrology," HDS-2, Second Edition, U.S. Department of Transportation, Washington D.C. (McCuen, R.H., Johnson, P.A., Ragan, R.M.)

Federal Highway Administration (FHWA), 2003, "User's Manual for FESWMS FST2DH, Two-dimensional depth-averaged Flow and Sediment Transport Model," Release 3, Report No. FHWA-RD-03-053, University of Kentucky Research Foundation, Lexington, KY (Froehlich, D.C.).

Federal Highway Administration (FHWA), 2008, "Highways in the Coastal Environment," HEC-25, Second Edition, Report No. FHWA-NHI-07-096 (Douglass, S.L., Krolak, J.).

Federal Highway Administration (FHWA), 2012, "Evaluating Scour at Bridges," HEC-18, Fifth Edition, Report No. FHWA-HIF-12-003 (Arneson, L.A., L.W. Zevenbergen, P.F. Lagasse, P.E. Clopper).

Federal Highway Administration (FHWA), 2012b, "Hydraulic Design of Safe Bridges," HDS-7, Report No. FHWA-HIF-12-018 (Zevenbergen, L.W., L.A. Arneson, J.H. Hunt, A.C. Miller).

Federal Highway Administration (FHWA), 2017, "A Primer on Modeling in the Coastal Environment," Report No. FHWA-HIF-18-002 (Webb, B.).

Froehlich, D.C., 1989, HW031.D - Finite Element Surface-Water Modeling System: Two-Dimensional Flow in a Horizontal Plane--Users Manual: Federal Highway Administration Report FHWA-RD-88-177.

International Hydrographic Organization (IHO), 2008, "IHO Standards for Hydrographic Surveys', 5th Edition, Special Publication No. 44. International Hydrographic Bureau MONACO

Jarrett, R.D., 1985: Determination of Roughness Coefficients for Streams in Colorado. USGS Water-Resources Investigation Report 85-4004.

Julien, P.Y., 2008, "Erosion and Sedimentation," Second Edition, Cambridge University Press.

Kramer, C.M., 2013, "Long Term Monitoring of Flow Deflection Structures in Northwest Rivers," River Restoration Northwest Conference, Skamania, WA.

Manning, R., 1891, "On the Flow of Water in Open Channels and Pipes," Transactions of the Institution of Civil Engineers of Ireland. Volume 20: pp 161-207.

National Cooperative Highway Research Program (NCHRP), 2006, "Criteria for Selecting Hydraulic Models,) Web-Only Document 106, Transportation Research Board (Gosselin, M., D.M. Sheppard, S. McLemore).

National Cooperative Highway Research Program (NCHRP), 2016, "Evaluation and Assessment of Environmentally Sensitive Stream Bank Protection Measures," Report 822 (Lagasse P.F., Clopper P.E., Thornton C.I., Shields, Jr. F.D., McCullah J., Spitz W.J.)

National Geodetic Survey (NGS), 2011, "The Effect of Modernizing the National Datums on Floodplain Mapping" (Youngman, M., Smith, D., Lokken, S., Langan, T.) In cooperation with the North Carolina Geodetic Survey, North Carolina Floodplain Mapping, and FEMA.

National Research Council of the National Academies (NRC), 2009, "Mapping the Zone, Improving Flood Map Accuracy," Committee on FEMA Flood Maps, Board on Earth Sciences and

Resources/Mapping Science Committee, Water and Science Technology Board, The National Academies Press, Washington, D.C.

Neumann (1947), "The Mathematician," in The Works of the Mind, edited by R. B. Heywood, University of Chicago Press, Chicago.

Sarrate, J., Palau, J. and Huerta, A. (2003), Numerical representation of the quality measures of triangles and triangular meshes. Communications in Numerical Methods in Engineering., 19: 551-561.

Simons, D.B. and Richardson, E.V. (1961). Forms of bed roughness in alluvial channels. Journal of Hydraulic Division of the American Society of Civil Engineers, v. 87(HY3), p. 87-105.

Timm, R. T., Caldwell, L., Nelson, A. D., Long, C., Chilibeck, M. B., Johnson, Mark, Ross, K., Muller, A., and Brown, J. (In Review) Drones, hydraulics, and climate change: Inferring barriers to steelhead spawning migrations. WIREs Water.

U.S. Army Corps of Engineers (USACE), 1991, " HEC-2 Water Surface Profiles, User's Manual," CEIWR-HEC

U.S. Army Corps of Engineers (USACE), 2016, "HEC-RAS River Analysis System, 2D Modeling User's Manual Version 5.0," (Brunner, G.W.)

U.S. Army Corps of Engineers (USACE), 2017, "Policies, Guidance, and Requirements for Hydrographic Surveying and Mapping Digital Products," ER 1110-2-8164

U.S. Department of Agriculture (USDA), 2001, "Sampling Surface and Subsurface Particle-Size Distributions in Wadable Gravel- and Cobble-Bed Streams for Analyses in Sediment Transport, Hydraulics, and Streambed Monitoring,' General Technical Report RMRS-GTR-74 (Bunte, K., Abt, S.)

U.S. Department of Agriculture (USDA), 2007, "Technical Supplement 13A, Guidelines for Sampling Bed Material," 210-VI-NEH, Part 654 National Engineering Handbook.

U.S. Geological Survey (USGS), 1967, "Roughness Characteristics of Natural Channels," USGS Water Supply Paper 1849 (Barnes, H.H. Jr.).

U.S. Geological Survey (USGS), 1970, " Determination of the Manning Coefficient From Measured Bed Roughness in Natural Channels," USGS Water Supply Paper 1898-B (Limerinos, J.T.)

U.S. Geological Survey (USGS), 1989, "Guide for Selecting Manning's Roughness Coefficients for Natural Channels and Flood Plains," Water Supply Paper 2339 (Arcement, Jr., G.J. and V.R. Schneider).

U.S. Geological Survey (USGS), 1998, "Estimation of Roughness Coefficients for Natural Stream Channels with Vegetation," Water Supply Paper 2441 (Coon, W.F.)

U.S. Geological Survey (USGS), 2006, "A Guide to the Proper Selection and Use of Federally Approved Sediment and Water-Quality Samplers," USGS Open File Report 2005-1087 (Broderick, E.D., and the Federal Interagency Sedimentation Project)

U.S. Geological Survey (USGS), 2016, "Identifying and preserving high-water mark data: U.S. Geological Survey Techniques and Methods," book 3, chap. A24, Koenig, T.A., Bruce, J.L., O'Connor, J.E., McGee, B.D., Holmes, R.R., Jr., Hollins, Ryan, Forbes, B.T., Kohn, M.S., Schellekens, M.F., Martin, Z.W., and Peppler, M.C.

Van Den Berg, J.H. (1995). Prediction of alluvial channel pattern of perennial rivers, Geomorphology, v. 12, p. 259-279.

Yang, C.T., (1972). Unit stream power and sediment transport. Journal of Hydraulic Division of the American Society of Civil Engineers, v.98 (HY10), p. 1805-1826.

Yochum, S., Bledsoe, B., 2010, " Flow Resistance Estimation in High-Gradient Streams," USDA Natural Resources Conservation Services & Colorado State University.

9.2. HYPERLINKS

Angle Representation Region –
https://www.xmswiki.com/wiki/SMS:ARR_Mesh_Quality_Assessment_Plot

Brigham Young University sediment transport database –
http://worldwater.byu.edu/app/index.php/sediment/home

Determination of the Manning Coefficient from Measured Bed Roughness in Natural Channels –
https://pubs.usgs.gov/wsp/1898b/report.pdf

Determination of Roughness Coefficients for Streams in Colorado –
 https://pubs.usgs.gov/wri/1985/4004/report.pdf

Estimation of Roughness Coefficients for Natural Stream Channels with Vegetation –
https://pubs.usgs.gov/wsp/2441/report.pdf

FEMA Flood Map Service Center – https://msc.fema.gov/portal/home

FEMA's Guidance for Flood Risk Analysis and Mapping, Vertical Datum Conversion –
https://www.fema.gov/media-library-data/1406747117296-744b6bd203c18ada4806ad4e90c18b81/Vertical_Datum_Conversion_Guidance_May_2014.pdf

FEMA's vertical accuracy requirements –
https://www.fema.gov/media-library-data/1469794589266-f404b39e73fa7a1c5ffe4447636634d4/Elevation_Guidance_May_2016.pdf

Floodplain management – https://www.fema.gov/floodplain-management-definition

Floodplain mitigation – https://www.fema.gov/what-mitigation

Floodways – https://www.fema.gov/floodway

Flow Resistance Estimation in High-Gradient Streams –
https://acwi.gov/sos/pubs/2ndJFIC/Contents/5E_Yochum_01_04_10_2_.pdf

Guide for Selecting Manning's Roughness Coefficients for Natural Channels and Flood Plains –
https://pubs.usgs.gov/wsp/2339/report.pdf

HEC-RAS River Analysis System Hydraulic Reference Manual – https://www.hec.usace.army.mil/software/hec-ras/documentation/HEC-RAS%205.0%20Reference%20Manual.pdf

Identifying and Preserving High-Water Mark Data (USGS 2016) – https://pubs.usgs.gov/tm/03/a24/tm3a24.pdf

International Hydrographic Organization standards (IHO 2008) – https://www.iho.int/iho_pubs/standard/S-44_5E.pdf

NAIP - https://www.fsa.usda.gov/programs-and-services/aerial-photography/imagery-programs/naip-imagery/

NCHRP 2006 – http://www.trb.org/Publications/Blurbs/159218.aspx

New datums – https://www.ngs.noaa.gov/datums/newdatums/index.shtml

NGS 2011 – https://www.ngs.noaa.gov/PUBS_LIB/Floodplain_Pilot_Project_Final.pdf?utm_medium=email&utm_source=GovDelivery

NGS Data Explorer – https://www.ngs.noaa.gov/NGSDataExplorer/

No-rise – https://www.fema.gov/no-rise-certification-floodways

NOAA Bathymetry – https://www.ngdc.noaa.gov/mgg/bathymetry/relief.html

NOAA – Hydrographic Survey Data – https://www.ngdc.noaa.gov/mgg/bathymetry/hydro.html

NOAA Hydrometeorological Design Studies Center – http://www.nws.noaa.gov/oh/hdsc/

NOAA River Observations – https://water.weather.gov/ahps/

Manning's n Values – http://www.fsl.orst.edu/geowater/FX3/help/8_Hydraulic_Reference/Mannings_n_Tables.htm

Positional Accuracy Standards for Digital Geospatial Data (ASPRS 2014) – https://www.asprs.org/a/society/committees/standards/Positional_Accuracy_Standards.pdf

The National Map – https://www.usgs.gov/core-science-systems/national-geospatial-program/national-map

Regulatory floodplain zones – https://www.fema.gov/flood-zones

Roughness Characteristics of Natural Channels – https://pubs.usgs.gov/wsp/wsp_1849/

sample 2D scope of work on FHWA hydraulics web page – https://www.fhwa.dot.gov/engineering/hydraulics/bridgehyd/bridge.cfm

table for Manning's n values – http://www.fsl.orst.edu/geowater/FX3/help/8_Hydraulic_Reference/Mannings_n_Tables.htm

Transient animation – https://www.xmswiki.com/wiki/SMS:Animations#Transient_Data_Animations

United States Interagency Elevation Inventory – https://www.coast.noaa.gov/inventory/

USACE requirements (USACE 2017)– https://www.publications.usace.army.mil/Portals/76/Publications/EngineerRegulations/ER_1110-2-8164.pdf?ver=2017-07-27-145155-727

USDA Data Gateway – https://datagateway.nrcs.usda.gov/

USDA Soil Maps – https://websoilsurvey.sc.egov.usda.gov/App/WebSoilSurvey.aspx

USDA Technical Supplement 13A, Guidelines for Sampling Bed Material (USDA 2007)– https://directives.sc.egov.usda.gov/OpenNonWebContent.aspx?content=17835.wba

USFS Sampling Surface and Subsurface Particle-Size Distributions in Wadable Gravel- and Cobble-Bed Streams for Analyses in Sediment Transport, Hydraulics, and Streambed Monitoring (USDA 2001)– https://www.fs.fed.us/rm/pubs/rmrs_gtr074.pdf

USGS Calibrated channel Manning's n Values – https://wwwrcamnl.wr.usgs.gov/sws/fieldmethods/Indirects/nvalues/index.htm

USGS – Bathymetric Surveys - https://www.usgs.gov/centers/oki-water/science/bathymetric-surveys?qt-science_center_objects=0#qt-science_center_objects

USGS Earth Explorer – https://earthexplorer.usgs.gov/

USGS Flood Event Viewer – https://stn.wim.usgs.gov/fev/

USGS Guide for Selecting Manning's Roughness Coefficients – https://pubs.usgs.gov/wsp/2339/report.pdf

USGS Hazards Data Distribution System – https://hddsexplorer.usgs.gov/

USGS National Streamflow Statistics Program – https://water.usgs.gov/osw/programs/nss/index.html

USGS StreamtStats – https://streamstats.usgs.gov/ss/

USGS Topographic Maps – https://ngmdb.usgs.gov/topoview/

USGS Water Information System – https://waterdata.usgs.gov/nwis/rt

9.3. IMAGE REFERENCES

Image | **Source**

Cover Image | Cover image by John Gussman

Figure 4.2 | The original maps are the copyright property of Google® Earth™ and can be accessed from https://www.google.com/earth (Google, 2019). The Authors developed the map overlays for this document.

Figure 4.3 | The original maps are the copyright property of Google® Earth™ and can be accessed from https://www.google.com/earth (Google, 2019). The Authors developed the map overlays for this document.

Figure 4.4 | The original maps are the copyright property of Google® Earth™ and can be accessed from https://www.google.com/earth (Google, 2019). The Authors developed the map overlays for this document.

Figure 4.5 | Photo credit – Dusty Robinson

Figure 4.7 | Image obtained from Hinds County Mississippi, 2D results produced from MDOT model

Figure 4.8 | Map from United States Interagency Elevation Inventory website; https://www.coast.noaa.gov/inventory/

Figure 4.10 | Image obtained from NAIP, and lidar data from Ayres Associates

Figure 4.11 | Lidar data obtained from Northern Colorado Geospatial Consortium, City of Fort Collins, CO

Figure 4.12 | Lidar data obtained from Northern Colorado Geospatial Consortium, City of Fort Collins, CO

Figure 4.13 | Image and photogrammetric data courtesy of the University of Iowa

Figure 4.15 | Point cloud data/image courtesy of Ayres Associates

Figure 4.16 | Image from Northern Colorado Geospatial Consortium, City of Loveland, CO

Figure 4.17 | Image obtained from USACE National Coastal Mapping Program

Figure 4.18 | Image obtained from NAIP and survey data courtesy of USACE

Figure 4.19 | Image obtained from NAIP and survey data courtesy of USACE

APPENDIX A

SRH-2D WITH SMS GRAPHICAL USER INTERFACE

CONTENTS

APPENDIX A

SRH-2D WITH SMS GRAPHICAL USER INTERFACE

A.1. OVERVIEW

There are many 2D models available that can be effectively used for riverine applications. However, regarding transportation hydraulics, few models currently include the capability to represent all features necessary for comprehensive analyses. Consequently, since 2013 FHWA has partnered with the U.S. Bureau of Reclamation to advance the development of the SRH-2D hydraulic model for transportation hydraulic applications. Furthermore, to improve the efficiency of the modeling process and to make it more accessible to new users, FHWA also has supported the development of a Graphical User Interface (GUI) for the SRH-2D model in the Surface-water Modeling System (SMS) software package. While the main body of this reference manual addresses guidelines for using 2D models in general, the purpose of this appendix is to provide specific information and resources for using the SMS interface to develop 2D models.

A.1.1. Using this Appendix

Appendix A provides a brief introduction to the development of an SRH-2D model using the SMS custom GUI. Additional details of all the features in SMS and step-by-step instructions are available in the online SMS Wiki and the SMS Tutorials.

Hyperlinks for key topics or phrases are provided throughout this reference manual, linking to detailed articles in the SMS Wiki or other reference documents. Keep in mind that links are sometimes changed. Should you encounter broken links, search for the topic or phrase in the SMS Wiki or a web browser search.

The following bullets provide a brief outline of the major sections of this appendix. Depending on your experience, you may wish to review the section most applicable to your needs rather than read through the entire document. Hyperlinks to each section are included:

- Software Overview – A brief description of the SMS and SRH-2D software, how they function together, and available resources.
- Using the SMS Interface – A description of the foundational features of SMS used for developing SRH-2D models.
- Steps to Create an SRH-2D Model in SMS – A detailed description of the steps for developing an SRH-2D model in SMS.
- Modeling Bridges and Other Structures – A description of how to add bridges, culverts, weirs, and other structures to an existing SRH-2D model.
- Model Calibration/Sensitivity Tools in SMS – A discussion of calibration and sensitivity tools in SMS.
- Postprocessing and Graphical Visualization Tools in SMS – A presentation of the advanced visualization capabilities and features in SMS.

A.2. SOFTWARE OVERVIEW

There are two components to the software used for developing SRH-2D hydraulic models: the SRH-2D numerical model and SMS, which acts as the user interface for SRH-2D. This approach combines the best of both worlds. The SRH-2D model provides a freely available, state-of-the-art numerical model for performing hydraulic computations. The SMS interface contributes an advanced platform for using digital elevation data, GIS data, and other information to provide tools for quickly setting up and running models, as well as tools for visualization of results. Using these two software tools together results in a flexible and efficient system for developing and presenting 2D models.

A.2.1. SRH-2D

SRH-2D (or Sedimentation and River Hydraulics – 2-Dimensional model) is a 2D hydraulic, sediment, and temperature model for riverine applications. The SRH-2D model has been in development since 2004 by Dr. Yong Lai at the U.S. Bureau of Reclamation. SRH-2D uses a flexible mesh consisting of a hybrid of quadrilateral and triangular elements. SRH-2D solves the 2D dynamic wave equations (depth-averaged St. Venant equations) and implements a very robust and stable finite volume solver incorporating an implicit scheme for time integration along with a seamless wetting-drying algorithm. The following list summarizes the key capabilities of SRH-2D:

- Steady and unsteady flows
- Subcritical, trans-critical, and supercritical flow
- Multiple inflow and outflow boundary conditions
- Internal boundary conditions (source/sink)
- Depth-variable roughness
- Bridge pressure flow with overtopping
- Bridge piers and other structures represented as "holes" in the mesh
- Blocked obstructions
- Culvert hydraulics (1D with HY-8 and 2D with bridge pressure flow)
- Gates
- Weirs
- Sediment transport

More information is available on the USBR webpage for SRH-2D. Detailed information regarding the assumptions and use of the model is available in the SRH-2D Reference Manual.

A.2.2. SMS

SMS is a GUI that has been developed over more than 20 years for several 2D models, including SRH-2D. A custom interface for SRH-2D within the SMS package provides an intuitive, comprehensive platform that aids users through the entire modeling process: from data gathering and editing to the review of model results and preparation of graphical presentation materials. A list of the primary SMS features and capabilities is provided below:

- Automatic coordinate projections
- Digital elevation data import in both vector and raster formats (e.g., XYZ, TIF)
- Data filtering and transformation (e.g., lidar data)
- Review, edit, and merging tools for multiple elevation datasets

- Breakline generation and editing tools
- Aerial imagery imports from online and other sources
- GIS/CAD data file import
- Automated mesh generation tools
- Mesh quality review tools
- Manning's roughness definition for polygonal areas
- Boundary condition definitions
- Monitoring coverage for tracking results and verifying stability and continuity
- Hydraulic structures (bridge pressure flow, culverts, weirs, gates, and piers)
- Run Queue to execute and monitor the progress of single or multiple SRH-2D simulations
- Automatic import of SRH-2D solution files
- Observation profile or cross section plots
- Custom contour and vector plots
- Data calculator for computing additional parameters from 2D results (e.g., riprap size)
- Dataset comparison tools (e.g., existing vs. proposed water surface elevations, velocities)
- Summary tables for minimum, maximum, or average cross section values
- CAD/ArcGIS/Google Earth (*.kmz) export features
- Zonal classification tool for highlighting zones of specified criteria (e.g., habitat values)
- Film loop visualizations and time-series animations
- Functional surface views (e.g., 3D terrain and water surface plots)
- FEMA floodplain reference tools (e.g., DFIRM data, flood hazard maps)
- Floodplain and floodway delineation tools
- Bridge scour evaluation tools

More information on these and other features can be found on the SMS Wiki.

A.2.3. Data Flow

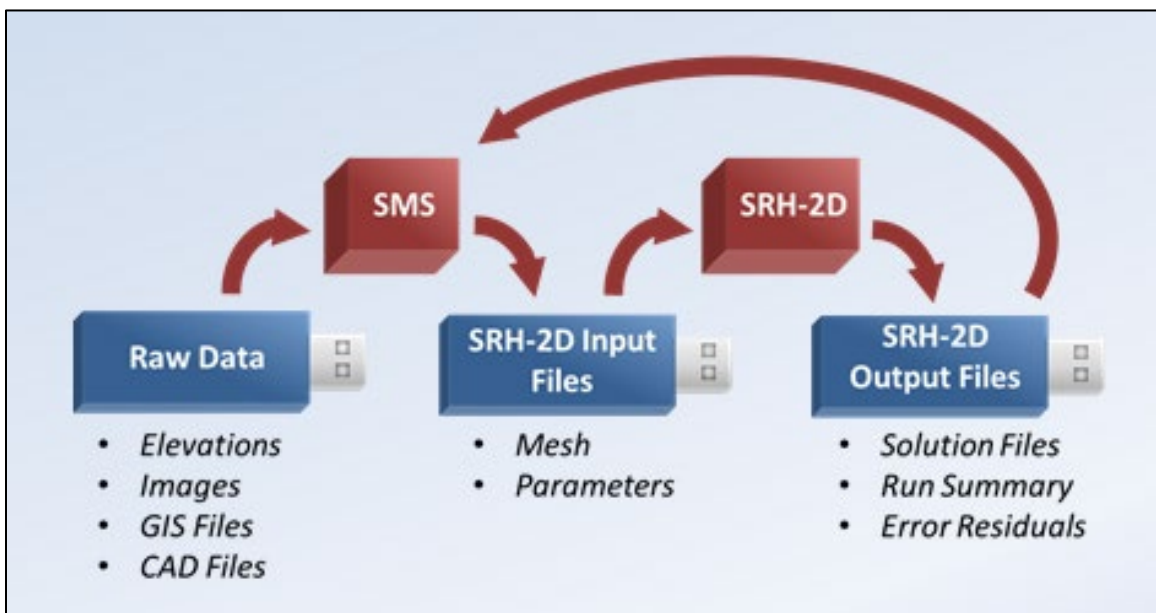Figure A.1 shows the data flow for developing and running models with SRH-2D using SMS:

Figure A.1. Data flow for developing and running SRH-2D models in SMS.

SMS can import raw digital elevation data needed for developing a 2D model along with data from other sources such as imagery, GIS data, or CAD data. Preprocessing tools in SMS are used to simplify and automate model development. SMS generates and writes SRH-2D input files for SRH-2D. SRH-2D is then executed using the standard input files created by SMS. The SRH-2D model runs within the SMS interface. After the model run has finished, the SRH-2D solution files are read into SMS, and the graphical post-processing tools in SMS are used to view and analyze the model results.

Numerous file types can be directly imported into SMS for use with SRH-2D. A non-exhaustive list can be found on the Non-native SMS Files wiki page.

A.2.4. Licensing

The SRH-2D model is in the public domain and is freely available for unrestricted use. The SMS interface is proprietary software, however, a community version is freely available that can be used to build, run, and review model results. Additional data analysis and graphical visualization tools are available in a commercial or "pro" version of SMS. Details of the capabilities and limitations of the Community Version of SMS can be found on the SMS Community Version wiki page.

SMS can be downloaded from the download page on the developer's website. The SMS installation installs both SMS and SRH-2D. When first launching SMS, a message appears to indicate no license is found, with options to Register SMS or use the Community Edition. If the Community Edition is selected, no further registration is required.

A license code is required to register the software. Those who purchase commercial licenses receive a license code with their purchase. For government users (FHWA or DOT), a license should be provided from FHWA Resource Team personnel. There is an option on the registration page to request a license. On the license request page (see Figure A.2) there are options to purchase a commercial license, request a government license, get an evaluation copy of the full version for 2 weeks, or request the community version.
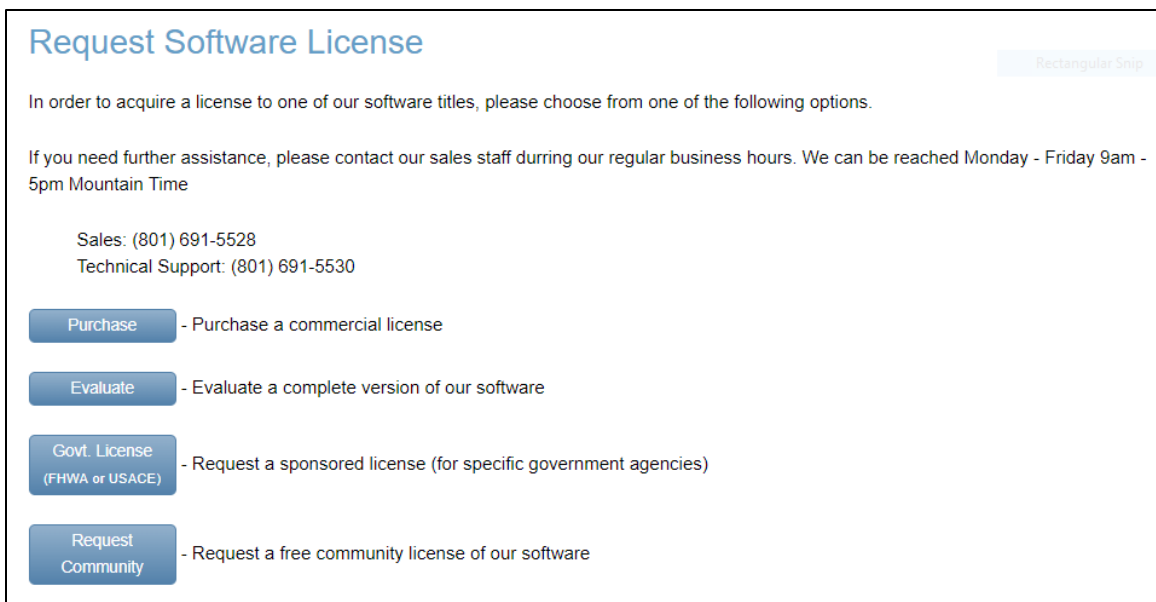

Request Software License

In order to acquire a license to one of our software titles, please choose from one of the following options.

If you need further assistance, please contact our sales staff durring our regular business hours. We can be reached Monday - Friday 9am - 5pm Mountain Time

Sales: (801) 691-5528
Technical Support: (801) 691-5530

Purchase - Purchase a commercial license

Evaluate - Evaluate a complete version of our software

Govt. License (FHWA or USACE) - Request a sponsored license (for specific government agencies)

Request Community - Request a free community license of our software

Figure A.2. SMS options to request a license.

FHWA currently provides "pro" licenses of SMS with the SRH-2D custom interface to all FHWA and state department of transportation (DOT) personnel. Other users may use the community version for free or purchase a commercial license. Licenses for FHWA and state DOT personnel are obtained through the FHWA Resource Center, while licenses for all others are obtained from Aquaveo.

A.2.5. Training, Learning, and Support

The FHWA National Highway Institute (NHI) offers multiple training opportunities for new users to learn 2D modeling and for others to sharpen their skills. The primary instructor-led course "Two-Dimensional Hydraulic Modeling of Rivers at Highway Encroachments" provides three days of comprehensive training on model development, execution, reviewing results, and graphical visualization summaries. State DOTs across the country regularly host this course. Some public seats are typically available, but anyone can host a course. Contact NHI for more information or questions about this course.

Two supplemental online web conference training courses are currently available: course 135095A – SRH-2D Model Data files, Diagnostics, & Verifying 2D Model Results, and course 135095B – Model Terrain Development with Various Data Sources. Each course consists of about four lessons typically covered on three separate days in two-three hour sessions. In addition, a modeling exercise is completed independently and reviewed during the last session. Additional training options are planned for the future. These courses are scheduled according to demand but may also be requested by anyone who has a group of at least eight participants.

The SMS Learning Center provides access to many options for users who wish to learn to use SMS and SRH-2D on their own, including videos, tutorials, and links to the SMS Wiki site and SMS user forum. The YouTube videos are about 5 to 10 minutes long and provide brief demonstrations of various SRH-2D modeling topics, while the tutorials include detailed annotated examples of many SRH-2D/SMS applications. Users may download the project files associated with each tutorial and work through the examples independently. For new and veteran users, the SMS Wiki site provides many helpful explanations of numerous modeling workflows, and more are regularly being added.

The FHWA resource center provides technical support for all FHWA and state DOT users. Others using the commercial version have access to technical support from Aquaveo. Technical support is not provided with the free community version.

A.3.   USING THE SMS INTERFACE

A.3.1. SMS Window Layout
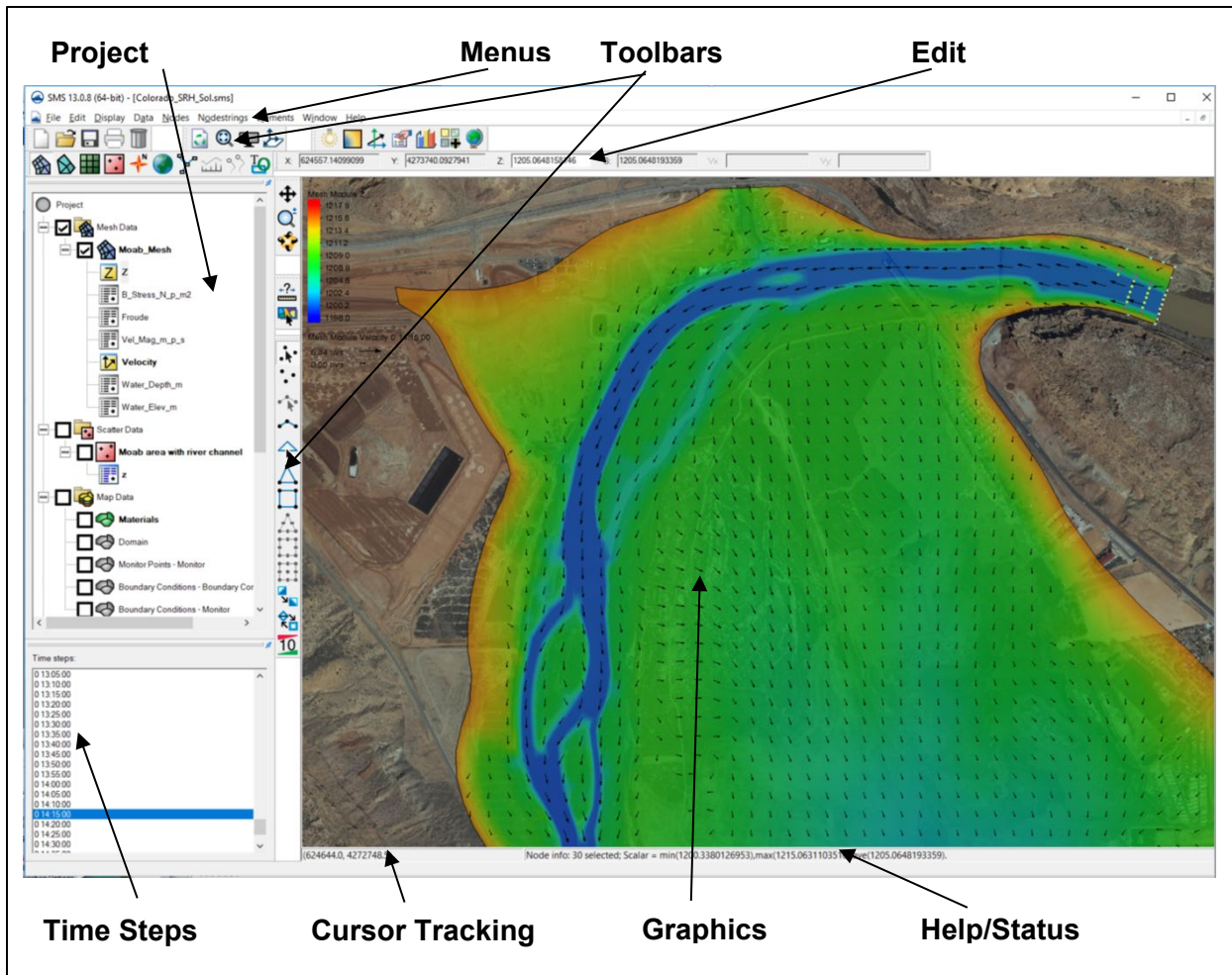
Figure A.3. shows the components of the SMS interface.

Figure A.3. SMS interface.

The SMS interface is a standard Windows-based interface with several different windows:

- Menus – Standard menu commands for SMS functions
- Graphics Window – Main window for visualizing and interacting with model data
- Toolbars – Tools for creating and selecting data objects, manipulating the display such as zoom, pan, rotate, and various other SMS functions
- Project Explorer – Display and management of all data objects loaded into SMS
- Edit Window – View, assign, or edit coordinates and dataset values of selected point objects
- Time Step Window – List of time steps for the model solution and control of the time step displayed
- Cursor Tracking – Displays the cursor location coordinates and dataset value at that location
- Help/Status Window - Displays help messages and information about selected data

The Toolbar includes commonly used tools divided into the following four sections:

- **Modules**: Different modules are used in SMS to operate on different data types. For SRH-2D modeling the Mesh, Scatter, and Map modules are commonly used. The Dynamic tools (see below) update based on the current module and model.
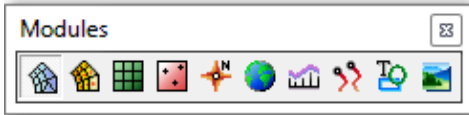

- **Static Tools***: Zoom* and *Pan* tools that control the portion of the objects displayed on the screen. The *Rotate* tool allows rotation to oblique views.
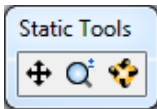
- **Dynamic Tools**: Module-specific tools. These tools are used to create, select, or modify objects for a given module. The icons with arrows indicate "select" tools, while those without arrows are "create" tools. When a dataset is selected, the appropriate module is activated, and the tools for that type of data are displayed. These tools allow for creating, editing, or performing functions for the types of data contained in the current module. The dynamic toolbar for the Map module is shown below as an example.

- **Macros:** Shortcuts for frequently used menu commands such as file open, save project, frame, refreshing the display, or set display options.
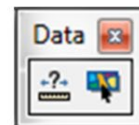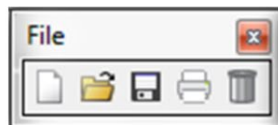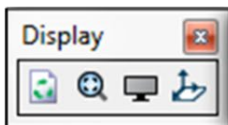
A.3.2. Project Explorer

The Project Explorer is a hierarchical view of all the data currently loaded into SMS. The data are organized in a "data tree" similar to CAD or GIS programs with data types grouped together. Checkboxes next to each dataset, or object, control whether the data it contains are displayed in the main window. Within each main folder, only one vector dataset and one scalar dataset are active at a time. To create, select, or edit data in a dataset or object, that dataset or object should be active. An SMS dataset or object is made active by left-clicking it in the Project Explorer. SMS indicates which dataset or object is currently active by making its text label bold and filling the icon to the left of the label with color. There are also many right-click functions available in the Project Explorer to perform various operations such as converting data, renaming datasets, specifying projection information, etc.

A.3.3. GIS-Based Model Development in SMS

Feature Objects

SMS uses a simplified GIS-based approach to facilitate the model development process. For this approach, points, arcs, and polygons (called "Feature Objects" in SMS) are created (see Figure A.4). The Map Module in SMS manages feature objects. It has specific tools for creating, editing, displaying, and deleting feature objects. Typically users digitize points or arcs to create feature objects using the create tools in the Map Module toolbar. Feature objects can also be created by converting CAD or GIS data by importing the data files then right-clicking on the data object and selecting the *Convert to Map* option.

Attributes are assigned to these Feature Objects to define model parameters such as boundary conditions and material properties. Feature polygons are also used to assign parameters for automated mesh generation. SMS then maps the parameters from feature objects to the SRH-2D mesh before exporting the SRH-2D input files. This method for creating model inputs simplifies the model development process and allows for greater flexibility when running multiple scenarios. Attributes are assigned to feature objects by double-clicking on the object using the appropriate selection tool or selecting the *Feature Object | Assign Attributes* command.

Feature Points are stand-alone x, y, z locations. Feature Nodes are points that form the endpoints of an arc. Feature Vertices are intermediate points on an arc that are only used to define the arc's geometry.

Feature Arcs are similar to polylines in other GIS and CAD programs and are the building blocks of polygons in SMS. Arcs are most often digitized directly in SMS. Endpoints of arcs are called nodes, and the intermediate points are called vertices. Alternately, arcs representing key planimetric features (i.e., pier locations, roadway alignments, bridge deck, etc.) can be created from data in CAD or GIS files. Typically, boundary conditions are assigned as attributes of a feature arc.

Feature Polygons are defined as one or more arcs that form a closed loop. The user should take care to be sure arcs connect properly to form closed polygons. Polygon attributes apply to the area inside the polygon. Material properties are assigned as attributes of Feature Polygons. Feature Polygon attributes are also used to assign parameters for automated mesh generation.
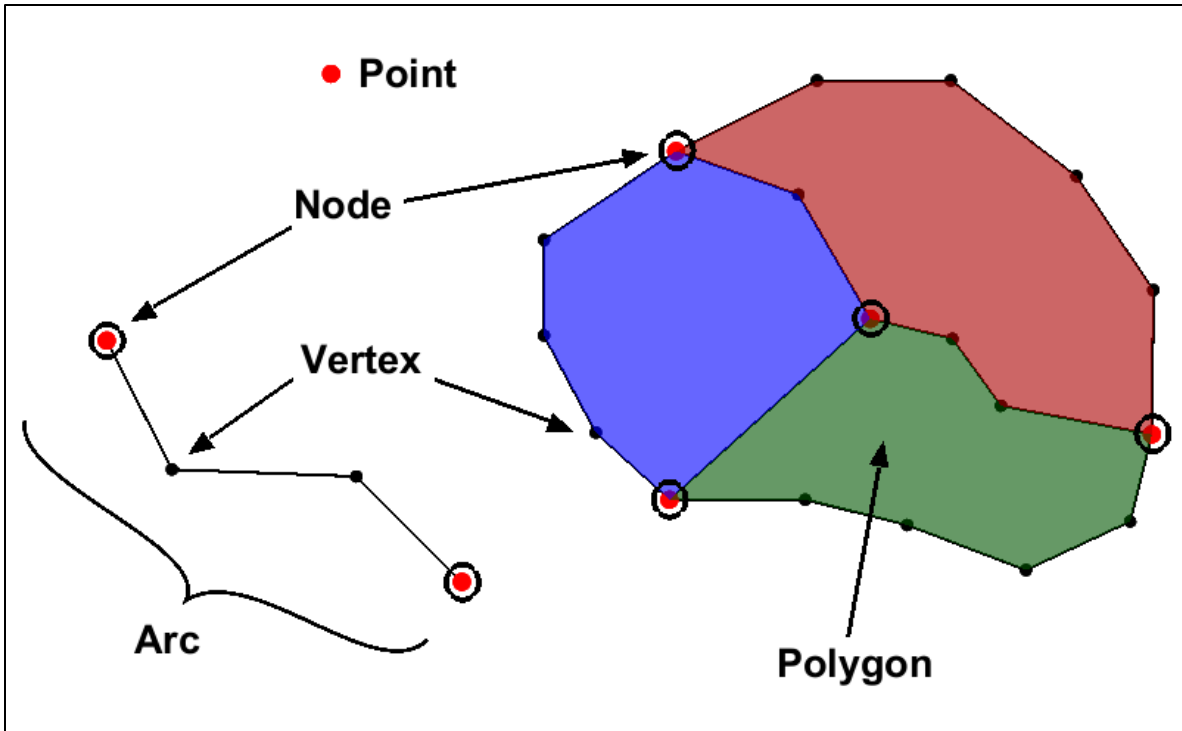
Figure A.4. Feature objects.

[Building Polygons](#)

Once all arcs have been created and connected to form closed areas, polygons should be built. This is done by executing the *Feature Objects | Build Polygons* menu command in the Map Module. No polygons exist until this command is executed. Once created, the user can verify that the polygons were built correctly by choosing the Select Feature Polygon ⬚ tool and clicking on a polygon. This highlights the selected polygon. Polygons can also be verified by turning on the *Fill Polygon* option in the Map section of the [Display Options](#) dialog. The user should be sure arcs are connected properly to form polygons. Anytime arcs are deleted or modified the *Build Polygons* command should be executed again.

Sometimes an arc does not snap to the next arc to close a polygon. The user may zoom in to identify these issues. The *Feature Objects | [Clean](#)* command has options to help address issues related to connecting feature objects.

[Coverages](#)

A "coverage" in SMS is a layer in the Map Module that contains Feature Objects. A coverage can be thought of as a layer of vector data. Unlike most GIS systems, a single coverage in SMS can contain a combination of points, arcs, and polygons. Coverages are created by right-clicking on the Map Data section in the Project Explorer and selecting *New Coverage*.

Each coverage has a type assigned to it that controls the attributes that are assigned to feature objects in that coverage and determines how feature objects in the coverage are used. The coverage type also determines which SMS dialogs is displayed when assigning attributes to feature objects in each coverage. It is important to make sure that the correct type is assigned when creating a new coverage.

Development of an SRH-2D simulation typically uses four types of coverages, each with its specific function. These include:

- Mesh Generator coverage is used in conjunction with an elevation source (scatter set or raster) to create a mesh
- Boundary Condition coverage defines the hydraulic conditions at the boundaries
- Materials coverage defines material types, attributes, and zones
- Monitor coverage defines locations to monitor the simulation, extract data and verify results

A.3.4. Display Options

A frequently used feature in SMS is the Display Options dialog. It is accessed in the Display drop-down menu at the top of SMS or in the right-click menu for each main folder in the *Project Explorer*. In the Display Options dialog, users specify which objects are to be displayed in the display window and control how they are displayed. This includes options for the display of contours, vectors, etc. There is a separate screen of display options for each module (data type), selected in the right windowpane in the dialog. Figure A.5 shows some of the Display Options for the Mesh Module in SMS:
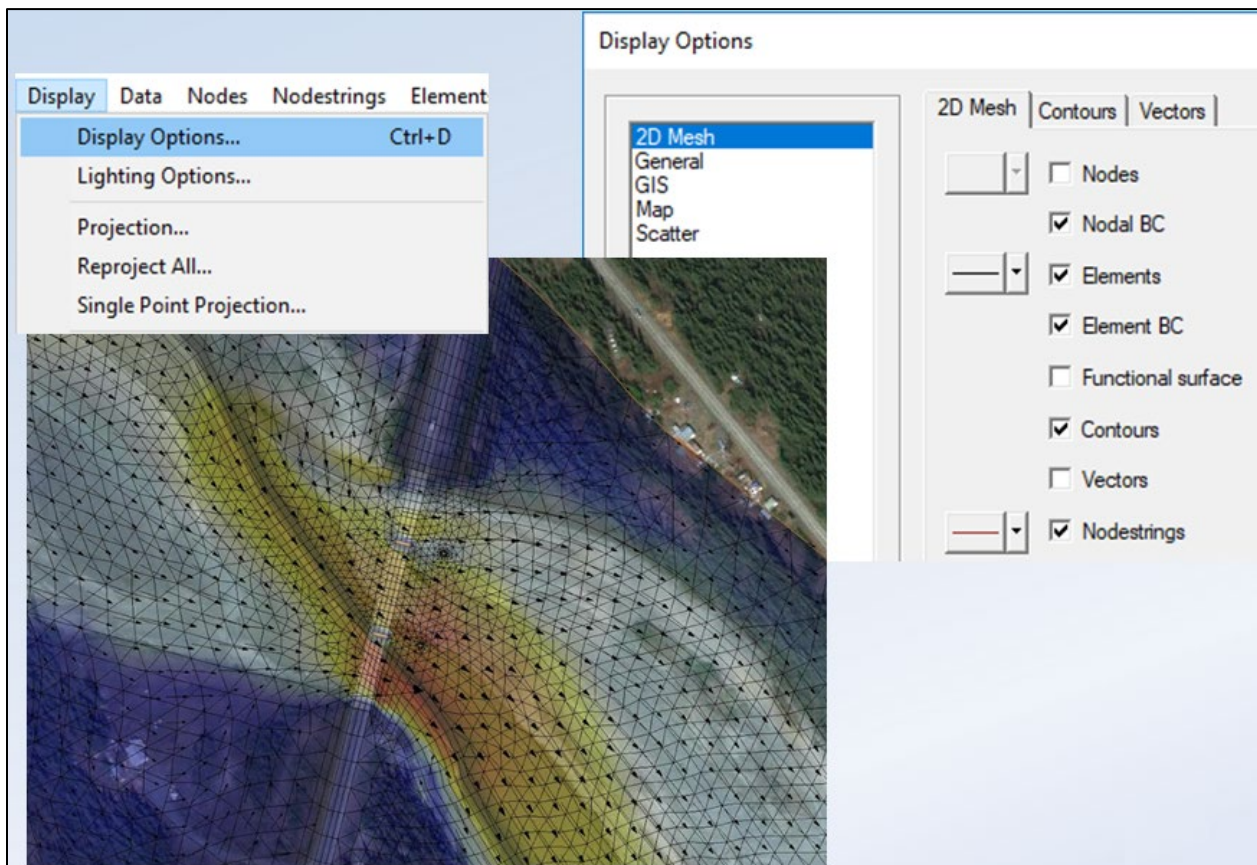

Figure A.5. Display options.

A.3.5. Selecting Data Objects

To select a specific type of object in SMS, the corresponding "Select" tool for that type of object should be used. These tools appear in the dynamic toolbar and are specific to the data type or module selected. "Select" tools' icons typically has an arrow pointing to an image of the object to be selected to distinguish them from "Create" tools icons. For example, the "Select" and "Create" tools for Feature Arcs are shown below:

- Create Feature Arc Tool –
- Select Feature Arc Tool –

Select objects by clicking on or within a "snapping" distance of the object. Objects are multi-selected by holding down the *Shift* key while clicking on additional objects. Other multi-select options available include dragging a box with the cursor, clicking out a polygon, or holding the *Ctrl* key and dragging the cursor to draw an arrow which selects objects the arrow intersects. When using SMS, remember that there is no "undo" function, so it is important to save your SMS project frequently.

A.4.   STEPS TO CREATE AN SRH-2D MODEL IN SMS

The steps for creating an SRH-2D model using SMS are consistent with the workflow presented in Figure 5.1. in Chapter 5 of this document. The first three steps and the 5th step (developing model domain) are generic and apply to all models, therefore, no additional instruction is provided in this appendix. The step for adding hydraulic structures is presented in detail in section A.5 of this appendix since this step usually occurs after an initial model is running successfully. The final step in the workflow, model review, is addressed in section A.7.

The remaining steps which are specific to developing SRH-2D models using SMS are presented below and provide a step-by-step guide for developing the model using SMS. These steps represent the "preprocessing" part of the modeling process, which involves developing and running an SRH-2D hydraulic model using the SMS interface:

- **Obtain Background Data**
- Generate a Mesh
- Specify Boundary Conditions
- Define Materials
- **Create Monitoring Points and Lines**
- Model Simulation
- Execute the Model

The **Background Data** includes terrain elevation data and other information used in defining the model. Four specific components in SMS are needed to run an SRH-2D model. These include mesh, boundary conditions, materials, and monitoring coverages. A fifth, optional component, may be added when blocked obstruction features are modeled. Generally, the **Mesh** is generated first, followed by coverages defining the **Boundary Conditions**, **Materials**, and **Monitoring** points and lines. Once these components are created, a **Simulation** is defined, and each of the components is then linked to the simulation. Finally, **Model Control** parameters are set, and the model is executed. The flowchart shown in Figure A.6 provides a graphical representation of the steps for developing an SRH-2D model using SMS. A detailed description of each of the steps in the workflow is presented below.
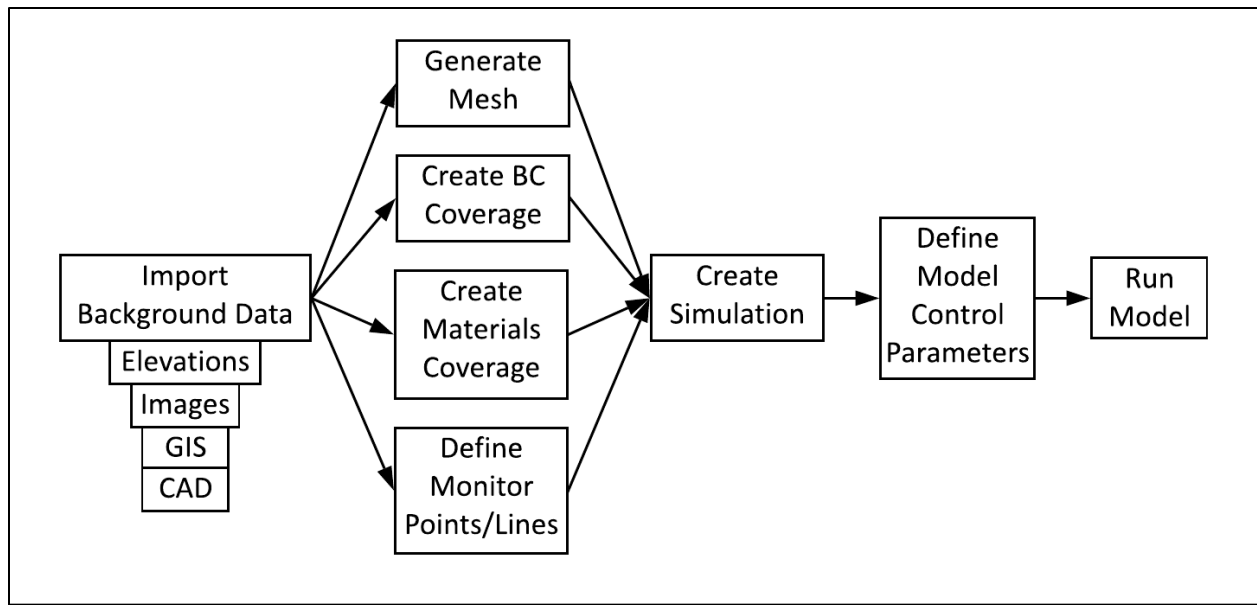
Figure A.6. SRH-2D model development workflow.

A.4.1. Importing Background Data

The typical steps for importing background data for developing a model are:

1. Set display projection
2. Import imagery
3. Import CAD or GIS Data (if needed)
4. Import digital elevation data
5. Merge elevation data
6. Review elevation data
7. Modify elevation surface

This section discusses each of the steps for importing background data and creating a surface that used to define model (mesh) elevations.

A.4.1.1. Coordinate Projections

An important consideration when importing data for use in SMS is that all data should be in a consistent coordinate system. The first step in developing a model in SMS is to specify the Display Projection which defines the coordinate system and units of measure for the model. This is done using the Display | Projection menu command, or SMS defaults to the first data layer read into the SMS project with a defined projection (.prj file).

All data should be in a consistent coordinate system to be used in SMS. SMS has the ability to convert data referenced to different projections "on the fly" to a common projection. Each data object read in can have its own assigned projection or coordinate system, provided it is a standard global coordinate system supported by SMS. Typically, the projection for each object is defined by an accompanying projection file (.prj) or has embedded projection information (such as for a GeoTIFF). If an object does not have an associated projection file, SMS assumes the data is in the display projection. If the data is not in the display projection, it will not align with other data

referenced. In this case, the data may seem to disappear—as if there are two "dots" of data widely separated. If this happens, the data without the projection information should be properly defined. This is done either externally or by right-clicking the data object, selecting the Projection option, and assigning the correct projection. Data can be reprojected and saved into a new projection if desired. Local projections can be used but cannot be combined with data from global projections as the conversion is not defined. Local projections are most often used for flume or research studies.

A.4.1.2. Importing Imagery

SMS supports the import of numerous image formats. Various sources of imagery can be found online. Most available images are geo-referenced and have a projection defined. To import an image, use the *File | Open* command or drag the file into the Display window in SMS. For images that are not geo-referenced, a dialog appears allowing you to register the image using the image registration tools in SMS.

Another way to import imagery is to load images from online imagery sources. This is done using the *File | Add Online Maps* menu command. This loads raster imagery using the window extents provided there is current data in the project. If not, a *Map Locator* window appears allowing the user to pan and zoom to the desired area. These maps are dynamic, meaning they refresh as you zoom or pan. This can be quite slow, so it is a best practice to load an image that covers the area for the entire project area then save a static image to the project directory.

A.4.1.3. Importing CAD and GIS Data

SMS also has the capability to import various types of CAD and GIS data. Any CAD or GIS data that has information defining or delineating key features within your project area can be helpful in developing a 2D model. Some examples include road centerline alignment or edges of the roadway, bridge features and culvert footprints, buildings or other structure footprints, floodplain limits, limits of vegetation, ordinary high-water mark definitions, and others. CAD or GIS data is imported using the *File | Open* command. These types of data are opened as separate objects in the Project Explorer.

Like images, these data are often used simply as reference data to help spatially orient the user in developing the model. However, CAD and GIS data often include elevation data that SMS can read and convert as well. It is most common to convert this data to create feature points, arcs, and polygons in a map coverage, but if the information includes elevation (z) data, it can be converted to a scatter set (terrain data) for use as a background data elevation set for mesh interpolation. This is done by right-clicking on the CAD or GIS object and selecting one of the Convert options.

A.4.1.4. Importing Digital Elevation Data

The most important data to bring into SMS for developing a 2D model is the digital elevation (terrain) data. The most common source for terrain data is lidar data. Other sources can include photogrammetry, hydrographic survey, or traditional field survey. Often data from multiple sources are needed to define the surface for a hydraulic model.

Elevation data in SMS is stored as scatter or raster data. Scatter and raster data are managed in the Scatter Module. Scatter datasets contain points with any x and y locations as well as associated dataset values that typically represent elevations. Other datasets can be stored in

scatter datasets, but for developing hydraulic models in SMS, scatter data usually refers to elevation data. Scatter data can also be triangulated to form a continuous surface (TIN). This surface is used as the elevation data source for assigning elevations to the model mesh in SMS. Elevation data is imported as scatter or raster data in several different ways, which are all accessed by using the *File | Open* command.

SMS supports importing several standard terrain data formats. Any data files with supported scatter data formats are opened using *File | Open*. SMS exports scatter files in an .h5 format, therefore, files previously exported from SMS are a format commonly used. LandXML data is also a supported data format for SMS. Microstation users can export TINs to LandXML format. Scatter data appears in the Project Explorer under the Scatter Data folder. Information on how to export data can be found on the [FHWA 2D Modeling](#) information page.

Elevation data from CAD stored on polyline points can be imported. TIN data from CAD triangles can also be imported. CAD data is converted to scatter data by right-clicking the CAD object in the Project Explorer and selecting the Convert options.

Hydrographic, field survey data, or other elevation data in a text delimited format (usually with x, y, and z values) is imported using the ["File Import Wizard"](#) in SMS, a tool similar to the Text Import Wizard in Microsoft Excel. When opening text files using the *File | Open* command, SMS provides the option to open using the File Import Wizard. Elevation data imported using this option is imported directly as a scatter dataset by specifying scatter data import in the dialog.

Lidar data points are imported in native .las or .laz file format. For these files, a lidar object appears in the GIS Data section of the Project Explorer. Right-clicking on a lidar object allows several different [lidar commands](#) for display options, converting to scatter data, interpolating to scatter set or raster, and merging lidar data.

Raster data, which can include lidar data, are gridded elevation data, often called DEM or DTM data. Rasters use less memory as only the elevation for each point along with an origin, delta x, and delta y is stored. SMS supports direct import using the *File | Open* command of several standard [raster formats](#) including .asc, .arc, .dem, .tif, .grd, and .flt. Third-party lidar processing software often exports elevation data in raster formats. Raster datasets appear as objects under GIS Data in the Project Explorer. Raster data can be used for assigning elevations to a mesh or can be converted to scatter data by right-clicking the raster data object and selecting one of the convert options.

Many sources for scatter data result in very large datasets that can cause performance issues when read into SMS. If you are having performance issues when attempting to import large datasets, the [filtering](#) tool in SMS can reduce the size of scatter datasets by eliminating points in areas of low elevation gradient. This tool is accessed using the *Data | Filter* command in the Scatter Module. Filtering is only available for scatter data sets, not for rasters.

A.4.1.5. Merging Elevation Datasets

Elevation datasets from multiple sources are often required to define a surface for a 2D model. For example, lidar data may be available for a site, but this data lacks any bathymetry data for the channel or detail beneath a bridge structure. Field survey data could be used in addition to the lidar data to define elevations in these locations. SMS has tools to merge multiple elevation datasets into a single scatter dataset. This tool is accessed using the *Scatter |* [Merge Sets](#) menu command.

In using the merge scatter sets tool, the user selects the scatter sets to be merged, then assigns a priority to each scatter set. Typically, the "Delete lower priority scatter points" option is selected, which deletes overlapping points form the lower priority set. Since SMS triangulates to the full extents of the scatter set, care should be taken to delete triangles in the higher priority scatter dataset that does not represent the surface so that overlapping lower priority elevation data points outside the extents of the higher priory dataset are not unintentionally deleted. For example, when merging stream cross sections with lidar data, the cross sections would be the higher priority dataset. In Figure A.7 the highlighted triangles for the cross section dataset would need to be deleted before merging so that underlying data points from the surrounding (lower priority) lidar data is not deleted during the merging process.
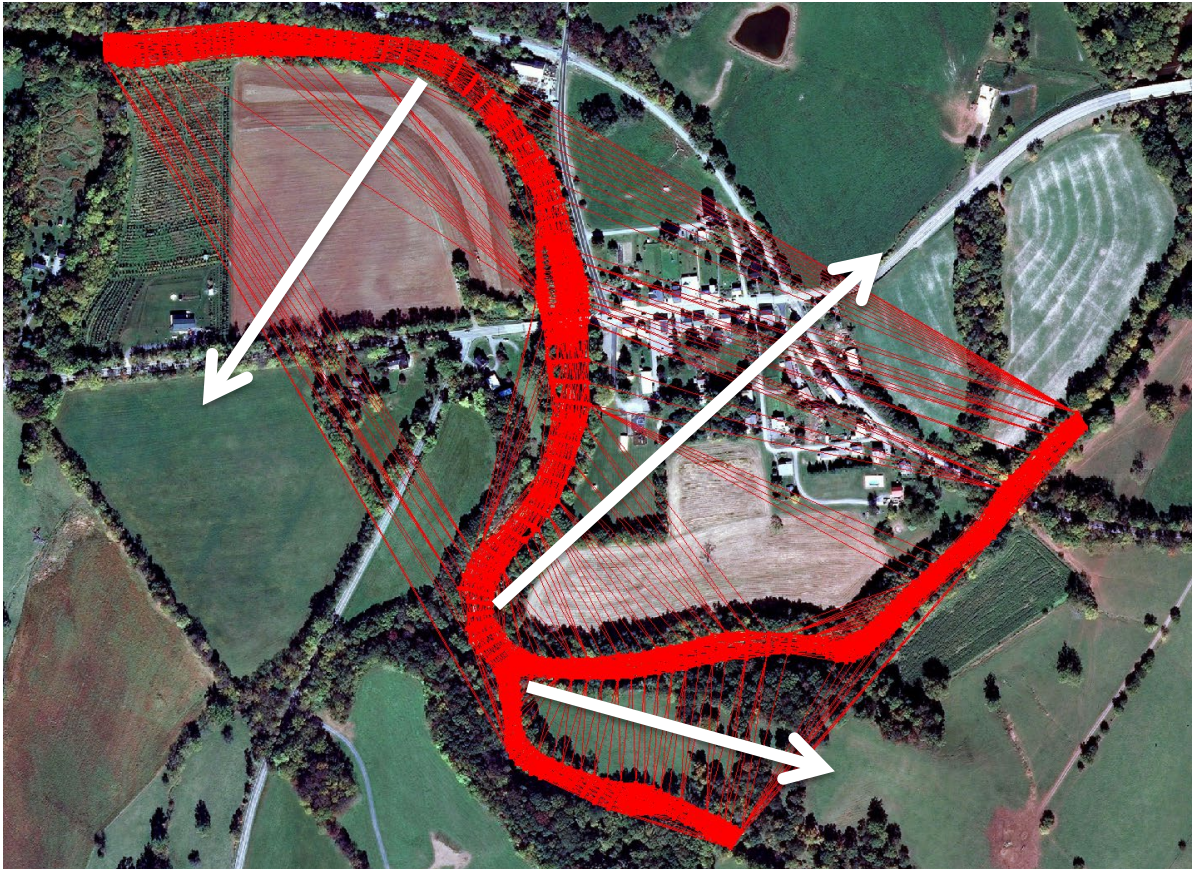
Figure A.7. Deleting triangles in SMS.

A.4.1.6. Reviewing Elevation Data

Elevation data should be reviewed to verify that the triangulated surface properly represents the site terrain. For example, inaccuracies in a model solution can arise from using a scatter set representing elevations with insufficient resolution or triangulation issues. Viewing contours of the surface can often help identify problems. Another method available is viewing the surface from an oblique 3D view in SMS with an exaggerated z-magnification. The Rotate ⬥ tool in SMS is used to view the elevation data from an oblique angle. SMS automatically sets a default z-magnification, but the user can increase the value in the General section of the *Display Options* dialog to exaggerate the z-values. Dynamically rotating the view can help better visualize the surface and identify problems.

A.4.1.7. Modifying Elevation Surfaces

If the review of the surface shows that triangulation or other issues result in poorly defined features (channels or ridges), the triangulation should be modified to represent the terrain better. Most triangulation issues are improved or corrected by adjusting the triangulation such that triangle edges better align with elevation contours. Two methods are available in SMS to modify a surface that has triangulation issues. The Swap Edges [icon] tool in SMS is used to swap an edge common to two adjacent triangles to connect the opposite corners. A more common way to align triangle edges is by adding breaklines. Breaklines are created by selecting the Create Scatter Breakline [icon] tool and connecting existing scatter data points, usually those that have similar elevations. Usually, multiple breaklines would be created to correct triangulation issues. Once breaklines are created, select the *Breaklines | Force Breaklines* menu command from the Breaklines menu to force triangle edges to coincide with the breaklines.

For example, in Figure A.8, an elevation dataset is displayed in a 3D oblique view. The left figure shows the default triangulation that creates false dams across portions of the channel. The center figure shows breaklines created to correct these issues, and the right figure shows the corrected surface with the breaklines forced into the triangulation.
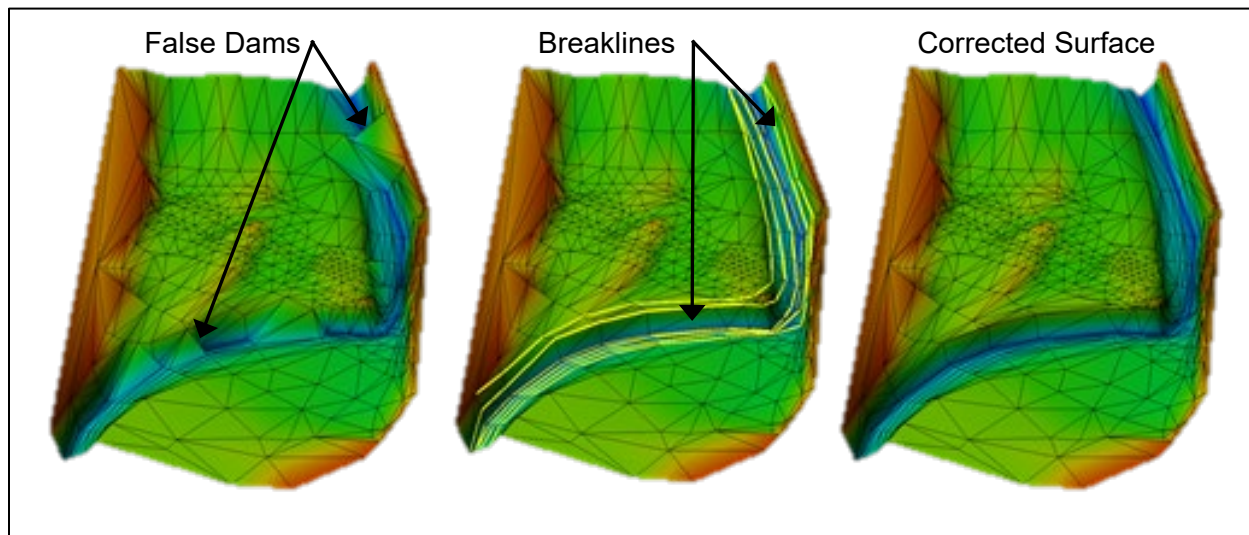

Figure A.8. Adding breaklines to correct triangulation issues.

A.4.1.8. Feature Stamping

Another way to modify the elevation dataset is to incorporate existing or proposed fill features, such as embankments or levees, which may not be part of the elevation dataset. This could also be true for proposed cut features such as a dredged channel or excavation, or perhaps to represent estimated conveyance when no bathymetry data is available. This would be done using the SMS Feature Stamping tools.

To create a feature stamp, a stamping coverage is first created by right-clicking Map Data in the Project Explorer and creating a new stamping coverage. Feature arcs are then added to the coverage at the location of the features to be stamped. Double-click on each feature to assign attributes defining the features. An elevation value is defined at vertices along the feature arc length. A cross section is defined at each vertex location. Typically, trapezoidal cross sections are defined using options in the dialog to set the top width and side slopes, then the trapezoidal cross

section is copied to be used for all vertex locations. Figure A.9 shows both cut and fill features stamped into an elevation surface.
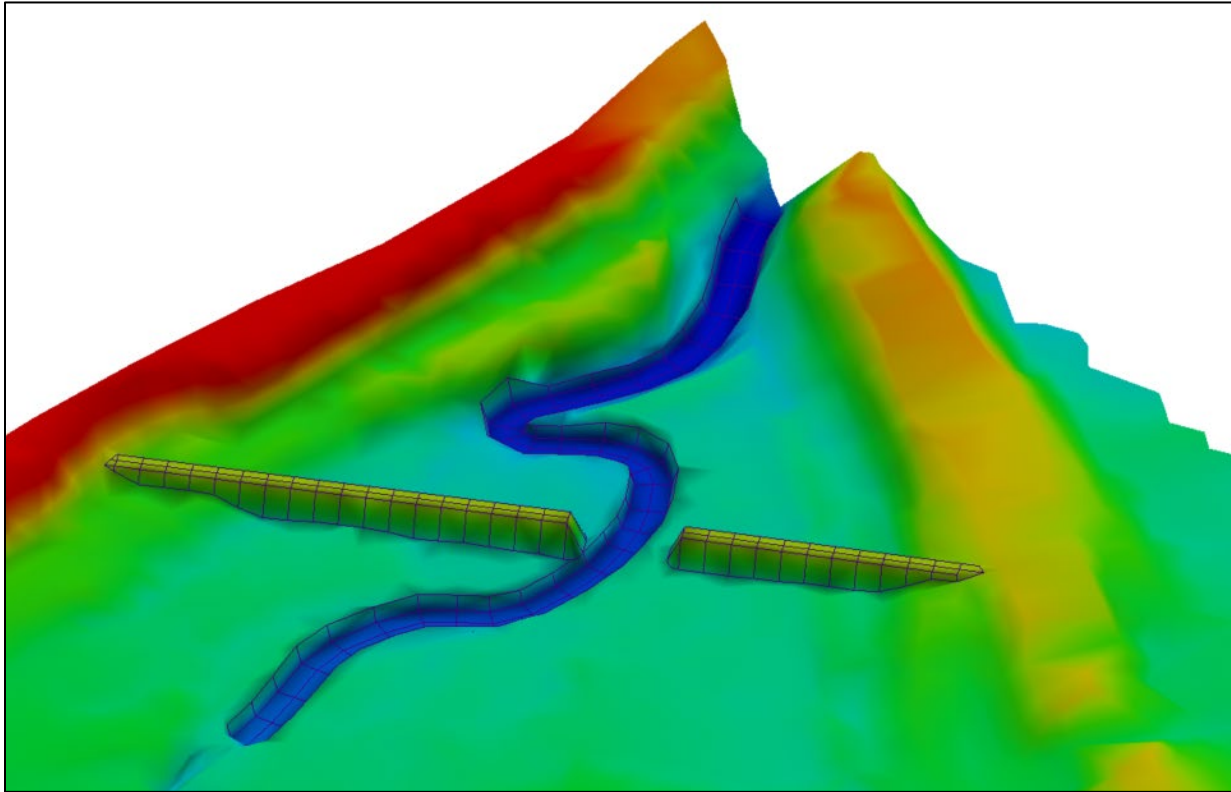

Figure A.9. Feature stamping examples.

A.4.2. Generating a Mesh

Once the background data is imported and a scatter set is defined representing the terrain of the project site, you are ready to move on to the next step in creating a model, which involves generating the mesh. The mesh is the most important component of a 2D model as it specifies the lateral and longitudinal limits to be modeled and serves as a representation of the underlying terrain to be used for the computational model (SRH-2D).

There are two basic strategies for creating a mesh.

- BASIC MESHING APPROACH: Single domain boundary with closely spaced vertices (Figure A. 10). The use of small elements helps to ensure that detail is captured to reflect hydraulic controls and significant changes in terrain. This approach requires relatively little effort, but the resulting mesh will likely have a large number of elements and longer simulation times.

- DIRECTED MESHING APPROACH: The use of added breaklines/polygons to represent hydraulic controls and areas of particular interest and generate a more efficient mesh (Figure A.11). This approach focuses on providing detail (smaller elements) where needed and larger elements elsewhere. It also aligns element edges along control features.
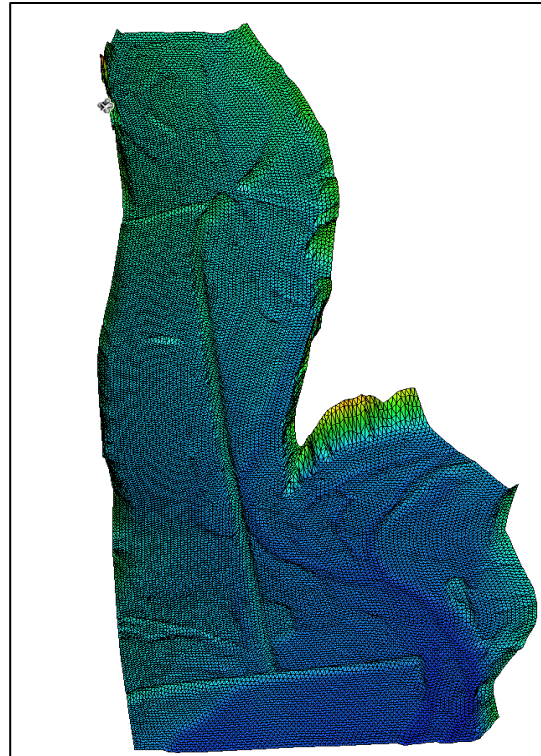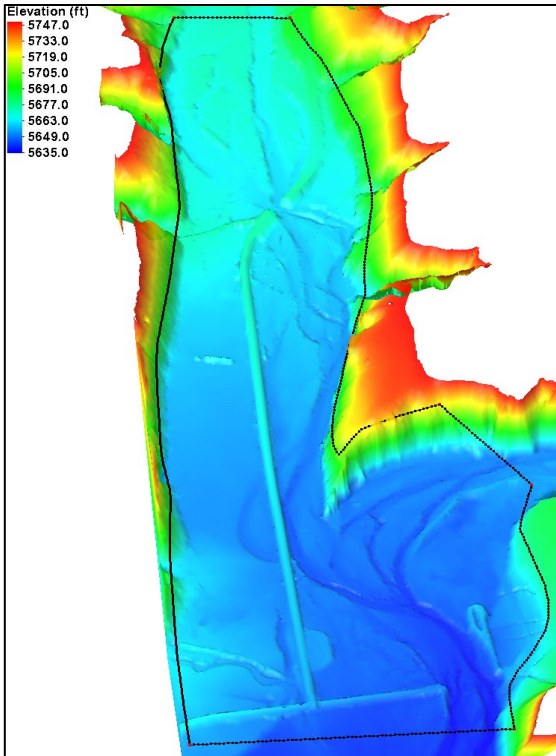
Figure A. 10. Terrain surface with single polygon (left) and resulting 42,000 element mesh with 15 ft vertex spacing (right).
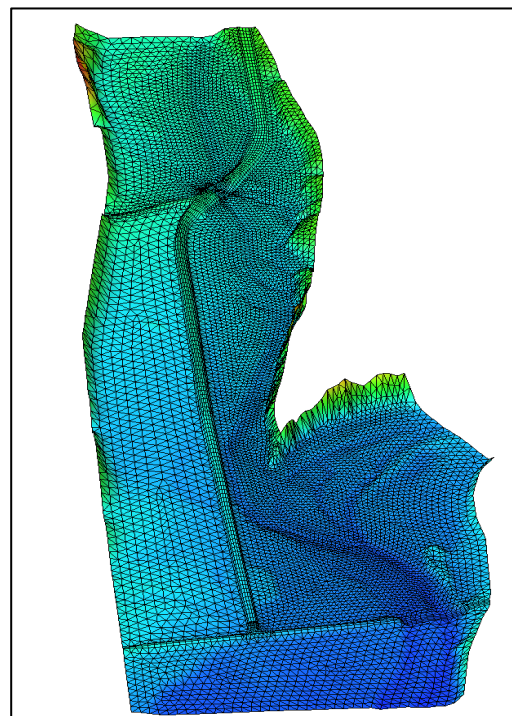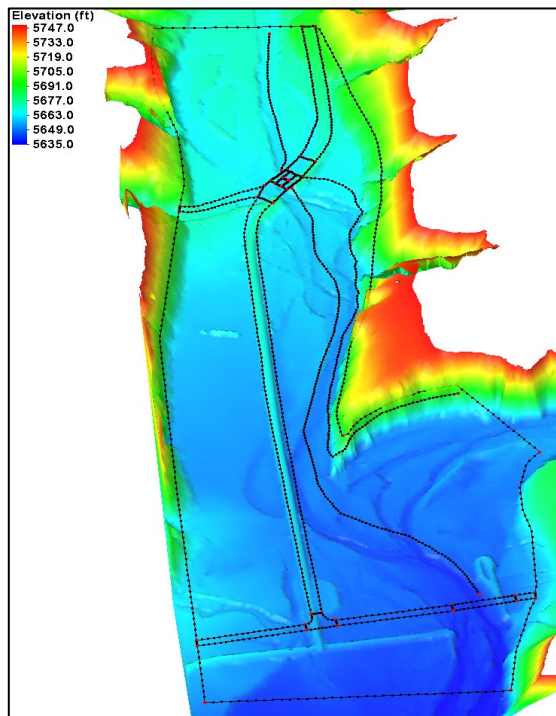

Figure A.11. Terrain surface with multiple polygons to align elements (left) and resulting 17,000 element mesh (right).

A.4.2.1. Creating the Mesh Generator Coverage

The first step in creating a mesh is to create a Mesh Generator coverage. This is done by right-clicking on the Map Data folder in the *Project Explorer*, selecting *New Coverage*, and setting the *Coverage type* to Mesh Generator. A different name for the coverage can be specified if desired.

A.4.2.2. Defining the Model Domain

Next, the model domain is defined in the Mesh Generator coverage. The model domain is defined by creating arcs to establish the perimeter of the area expected to be wet. Usually, this begins by digitizing arcs using the Create Feature Arc tool 📐. Arcs can also be created by converting lines from imported CAD or GIS data. Arcs are usually created near the locations where the upstream and downstream model boundary conditions are specified. The lateral model extents are then defined by creating arcs outside the expected inundation limits of the largest flow to be modeled to connect the upstream and downstream arcs creating a closed polygon.

A.4.2.3. Adding Internal Features

Once the extents of the domain are defined, arcs may be added to define internal features. These arcs may define channel banks or capture the details at bridges or other features. Once all the internal features are defined, the *Feature Objects | Build Polygons* menu command should be executed to create polygons. Check the polygons to be sure the arcs have been properly connected. This step is not required if using the simplified approach where a dense, uniform mesh is specified for the entire domain.

A.4.2.4. Setting Initial Vertex Spacing

Element resolution (size) is controlled by vertex spacing along arcs in the Mesh Generator coverage. When the mesh is created, the polygon is filled with elements, and the mesh generator automatically transitions the size of the elements between arcs of different vertex spacing.

Vertex spacing established when creating arcs is not typically uniform, it is standard practice to redistribute vertices for all arcs in the model to a chosen vertex spacing before assigning specific parameters for each polygon. This sets the resolution for the mesh. As parameters are defined for each polygon, vertex spacing can be modified to provide more or less resolution as needed. To set vertex spacing for all arcs, select all arcs and execute the *Feature Objects | Redistribute Vertices* menu command, then specify a spacing equal to the element size desired.

A.4.2.5. Assigning Meshing Parameters

After the Polygons are defined, automated mesh generation attributes are assigned to each polygon in the domain. This is done by selecting a polygon and executing the *Feature Objects | Attributes* menu command (or by double-clicking on the polygon). This brings up the 2D Mesh Polygon Properties dialog, as shown in Figure A.12.
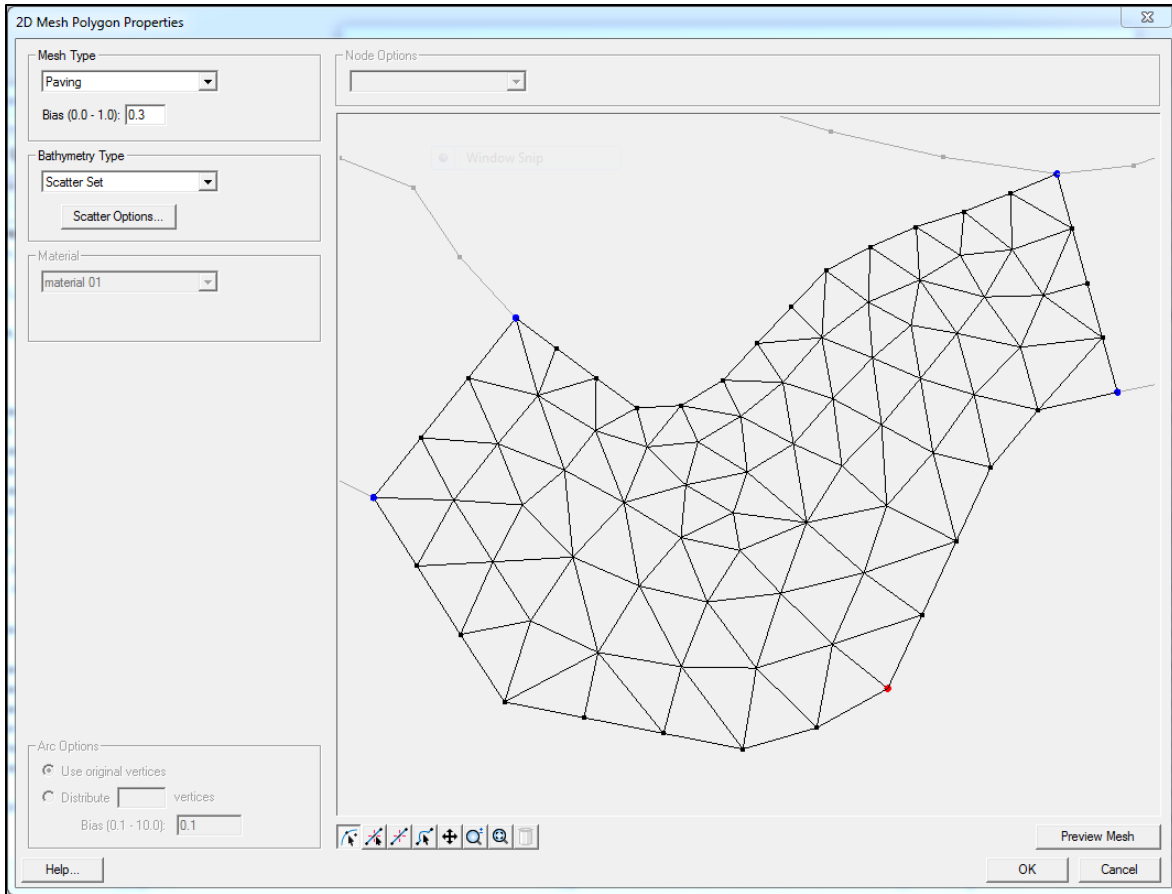
Figure A.12. 2D mesh polygon properties dialog showing paving option.

The mesh type should be specified first. The default mesh type is "Paving" that fills the polygon with triangles based on the vertex spacing along the arcs. "Paving" is generally the most widely used mesh type.

The other mesh type often used is "Patch." The "Patch" mesh type works best for long, narrow polygon areas. Therefore, it is often used to define channel or roadway areas.

The patch mesh type creates quadrilateral elements in a "checkerboard" pattern by connecting element edges between opposite sides. Triangular elements will be created to transition if there are an unequal number of elements on opposite sides. A quadrilateral mesh is more efficient and help to reduce the number of elements and simulation time. However, it does involve more time to build the model.

Quadrilateral patches need four distinct feature arcs to define the opposite "sides" of the patch. If there are more than four arcs, arcs can be merged in the *2D Mesh Polygon Properties* dialog shown in Figure A.13. Merging arcs for patching is done by selecting the node where the arc is to be merged and selecting "Merge" in the *Node Options* drop-down box. Splitting nodes has the reverse effect.
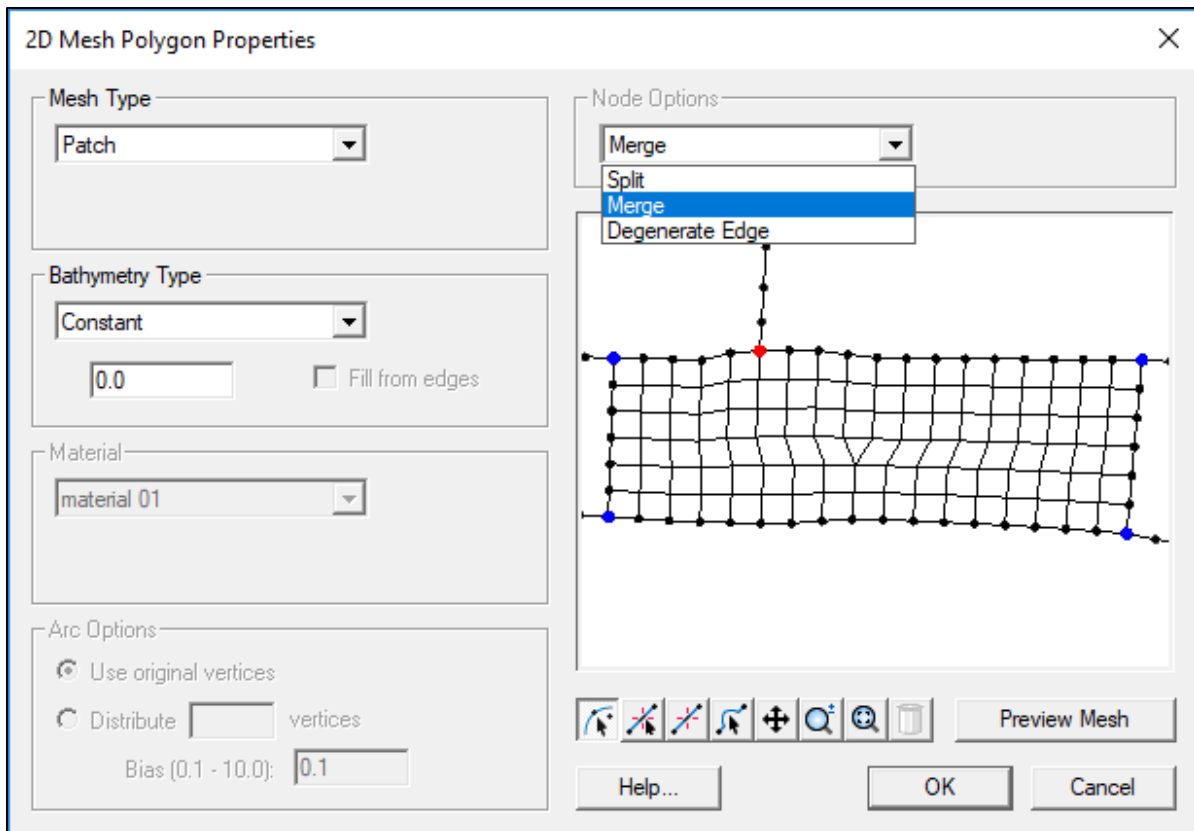
Figure A.13. 2D mesh polygon properties showing patch option.

The "None" mesh type option may be used to specify a polygon representing a hole in the mesh where no elements exist. There are other mesh types, but they are rarely used.

On the right side of the dialog, a display window shows a preview of how the mesh will be created based on the mesh type selected and the vertex spacing of the arcs. Be sure to use the Preview Mesh button to view the mesh, as using the *OK* button results in exiting the dialog and saving the vertex spacing and meshing parameters.

The *2D Mesh Polygon Properties* dialog also has tools for modifying the vertex distribution to adjust mesh density. The *Arc Options* section allows for redistribution of vertices within the dialog. The number of vertices is specified for selected arcs. A bias is specified for the vertex spacing (i.e., a bias of 2.0 means that the spacing of the vertices at the end of the arc is twice the spacing at the beginning).

The toolbar beneath the Preview window has identical Map Module tools to those located in the main window for selecting feature objects. This allows for making changes without exiting to the main window. The Create Vertex tool is used to adjust vertex spacing by adding vertices to arcs in the preview window. Vertices can also be selected and deleted.

The *Select Node* tool is used to select nodes to be specified as "Merge" or "Split" nodes using the *Node Options* section of the dialog. "Merge" nodes are usually used for merging arcs on polygons specified with a "Patch" mesh type. There are also Pan, Zoom, and Frame tools included in the dialog to allow for viewing polygons with larger or complex meshes.

Finally, a *Bathymetry Type* is selected which refers to the background terrain data set which used to define the elevations of the mesh. *Constant* at a value of 0.0 is the default. Typically, *Scatter Set* or *Raster is selected*. With the scatter set option selected, the *Scatter Options* button should be selected which brings up the Interpolation dialog. In this dialog, the scatter or raster set to be used for interpolation of elevations for the mesh nodes is specified.
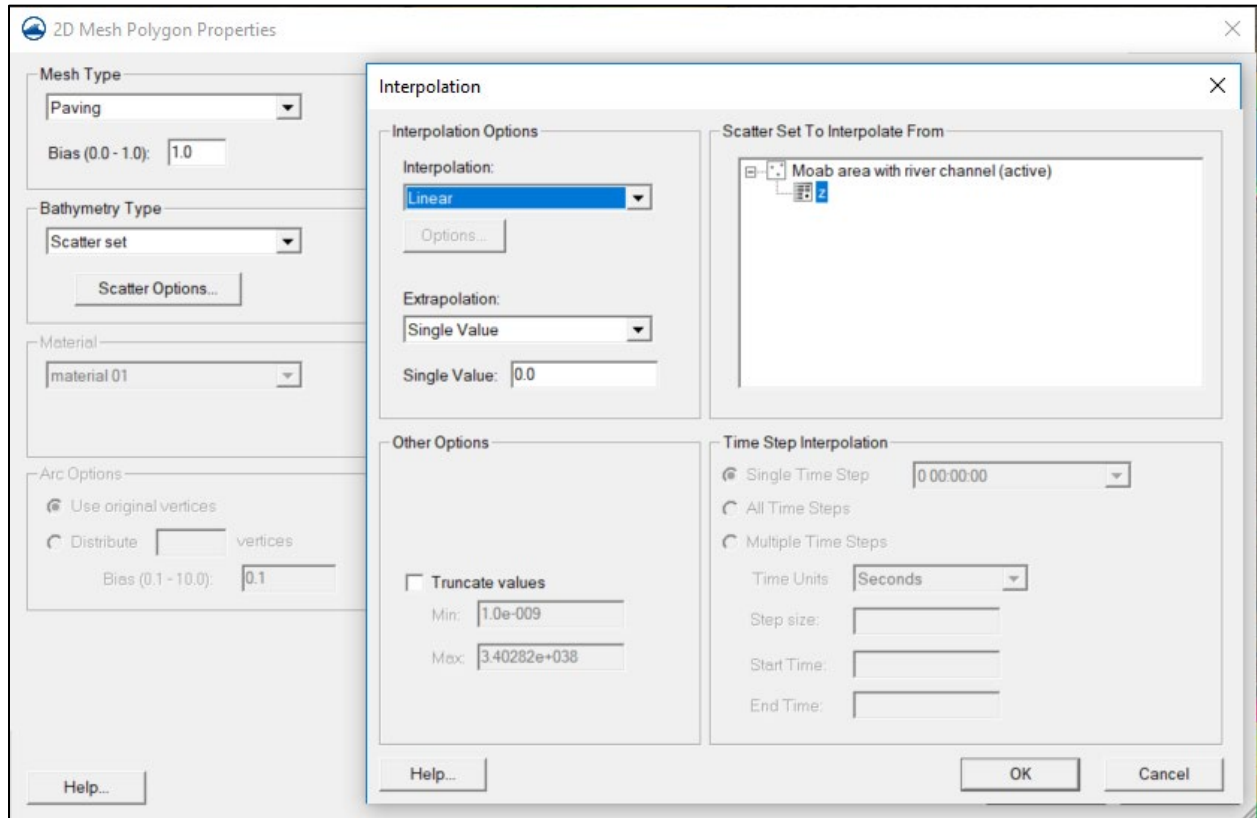

Figure A.14. 2D mesh polygon properties showing bathymetry type selection.

This process is repeated for each polygon in the model domain. Warning: Keep in mind that when changing vertex spacing on arcs common to adjacent polygons, the spacing is also changed for previously completed polygons changing how the mesh is created in those polygons.

A.4.2.6. Generating the Mesh

Once the mesh parameters and vertex spacing are set for all polygons, the mesh is ready to be generated using the *Feature Objects | Map -> 2D Mesh* menu command. This can also be done by right-clicking on the Mesh Generator coverage in the Project Explorer and selecting *Convert | Map -> 2D Mesh*. When the mesh generation is complete, a prompt appears asking the user to specify a name for the mesh.

When creating a mesh, it is best to start simple. It is good practice to begin by creating and running a simple mesh with a single polygon meshed using the "Paving" mesh type at a single element resolution. Detail can be added to the mesh to specify higher resolution, internal details, and structures for subsequent runs. For most SRH-2D projects at bridge crossings, an efficient, effective mesh should typically be less than 100,000 elements and commonly less than 50,000 elements.

Another factor to consider in specifying mesh resolution is that SRH-2D can become unstable if the vertical (z) dimension for the element becomes large in relation to its planar (x,y) dimensions. Limiting the element length to be equal to or greater than the flow depth can help to minimize/avoid modeling instabilities. Smaller elements in the immediate vicinity of structures (e.g., piers) are generally acceptable.

A.4.2.7. Review the Mesh

The mesh should always be reviewed to be sure it was created as desired, elevations were properly assigned, the resolution chosen results in mesh elevations that reasonably represent the underlying terrain surface, and the mesh quality is reasonable. To do this, the mesh elements and elevation contours should be turned on in the *Mesh* section of the *Display Options*. To better review the mesh, it is often best to turn off all other options and disable the display of scatter sets, coverages, and any images.

The *Display Options* dialog also has a Mesh Quality option that highlights areas of the model that violate mesh quality guidelines (see Figure A.15). Problem areas are highlighted with color-coded symbols. The Mesh Quality options are only guidelines, and having an area highlighted with one of the mesh quality categories does not always necessitate further action on the part of the user. Many of the highlighted areas will not result in model inaccuracies or instabilities, particularly if problem elements are not in areas of high elevation or velocity gradients. These problems would only need to be addressed if there are a large number of extreme cases of violation of the guidelines (similar to reasonably minimizing the warnings and notes in HEC-RAS). Otherwise, proceed with running the model and review to see if the model runs and seems to provide reasonable results.
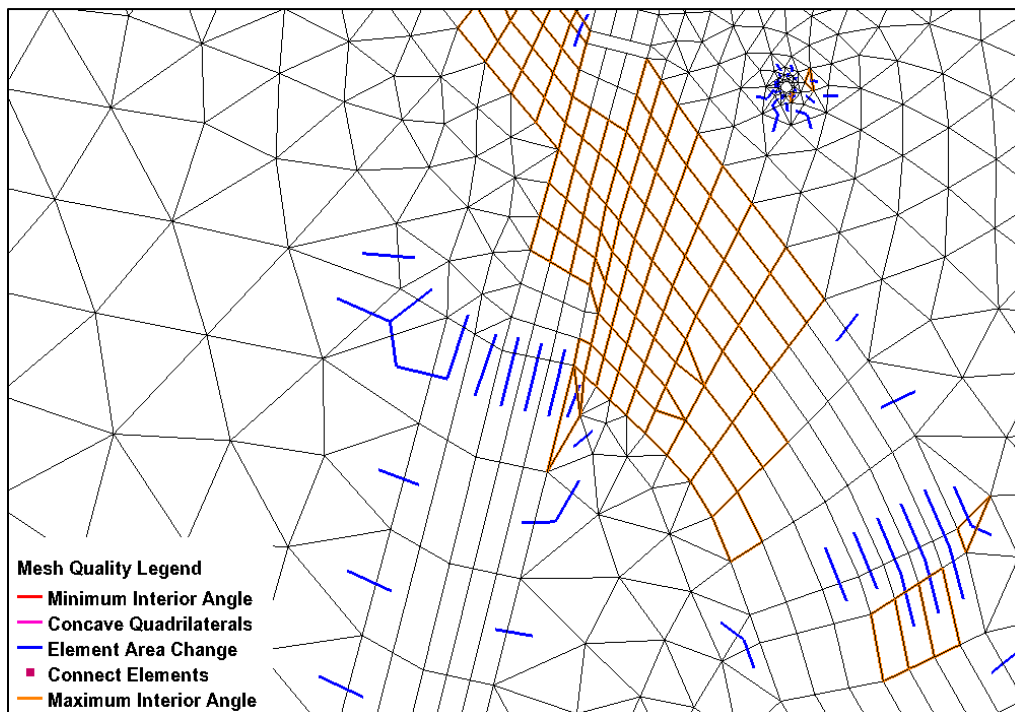

Mesh Quality Legend
— **Minimum Interior Angle**
— **Concave Quadrilaterals**
— **Element Area Change**
▪ **Connect Elements**
— **Maximum Interior Angle**

Figure A.15. Example of the effect of using a bias when redistributing.

A.4.3. Applying Boundary Conditions

The next step is to define the boundary conditions for the model. This is done by creating a Boundary Conditions coverage and specifying attributes to define the boundary conditions. The steps for creating model boundary conditions in SMS for an SRH-2D model are detailed in the sections below.

A.4.3.1. Creating the Boundary Condition Coverage

First, create a new SRH-2D Boundary Conditions coverage by right-clicking on the Map Data folder in the *Project Explorer* and selecting *New Coverage*. This brings up the *New Coverage* window where the coverage type is specified. Select the *SRH-2D | Boundary Condition* coverage type in this dialog to make it an SRH-2D Boundary Conditions coverage. A different name can be specified if desired.

A.4.3.2. Creating Boundary Condition Arcs

Arcs are created in this coverage to define boundary conditions (BCs) for the model. Usually, it is best to have the mesh visible when creating BC arcs. Arcs are typically placed near the upstream and downstream model extents, but they do not necessarily need to be coincident. Boundary conditions are mapped (snapped) to the mesh element edges nearest to each BC arc.

A.4.3.3. Defining Boundary Conditions

Boundary condition parameters are defined by selecting a BC arc, right-clicking on the arc, and selecting *Assign Linear BC*. This brings up the SRH-2D Linear BC dialog for specifying the boundary condition type and associated parameter values (see Figure A.16).

Inflow boundary condition types are used to specify open boundaries where water enters the model. The recommended inflow boundary condition type is "Inlet-Q," which is a specified subcritical inflow. The inflow is specified as either constant or Time Series (hydrograph). Usually, *Conveyance* is selected for the *Distribution at Inlet.* Supercritical inflow boundary conditions can be specified but are not recommended.

Outflow boundary condition types are used to specify open boundaries where water exits the model. The recommended outflow boundary condition type is usually "Exit-H," which is a specified water level. The water level is defined in one of three ways:

- Constant – a constant water level is specified for the arc
- Time series – the water level is specified as a function of time
- Rating curve – the water level is dependent on the flow rate exiting the model at the boundary based on a user-specified rating curve

By default, the model runs in *Unsteady* mode. When running a steady-state simulation, it is recommended to use the SRH-2D *Steady-State* option. To access this, right-click on the BC coverage, select *BC Types*, and turn the *Unsteady Output* off. It is acceptable to run in *Unsteady* mode, but the *Steady State* mode has been optimized to run more efficiently when only the final result is needed.
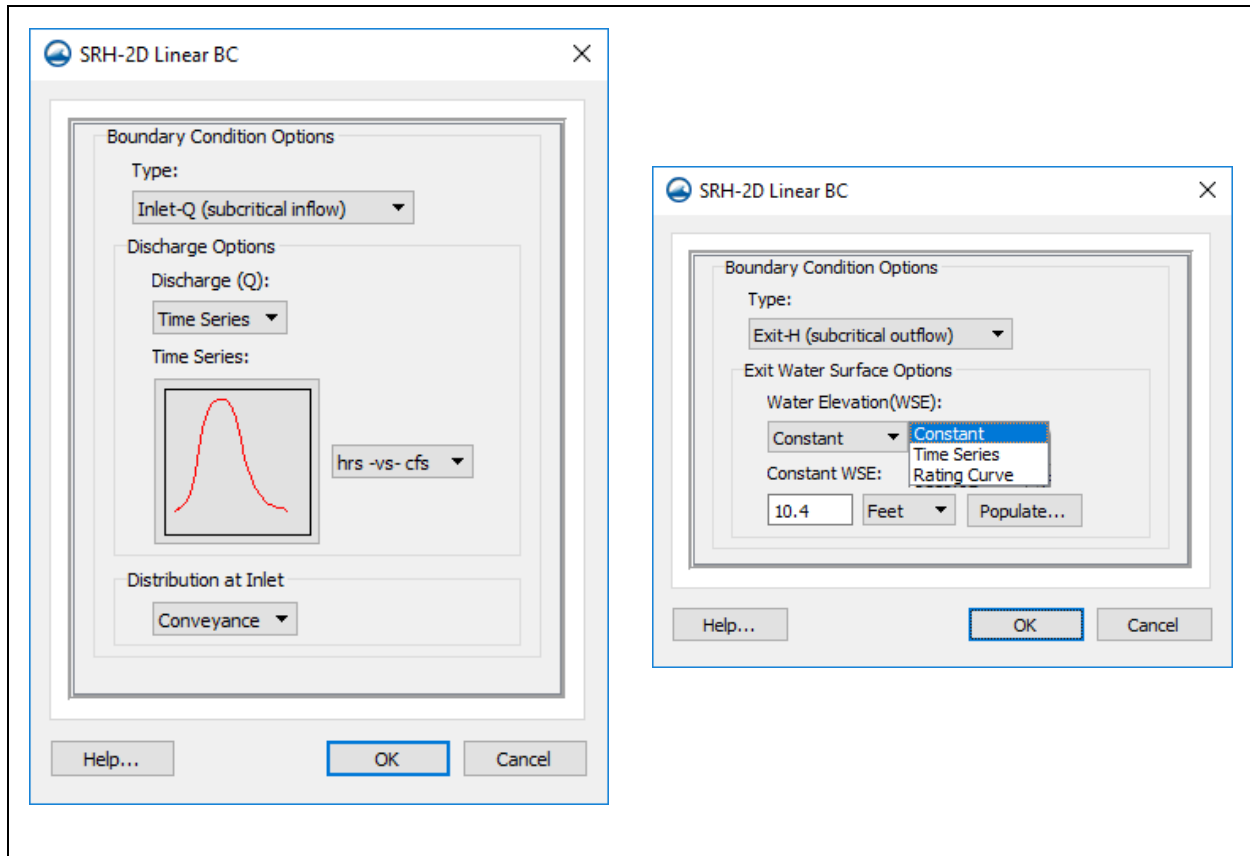
Figure A.16. SRH-2D linear BC dialogs.

Exit-Q is a specified flow rate at the outflow boundary. This is normally only used when a known amount of flow is being removed say at a diversion or lateral weir.

Supercritical outflow boundaries can be specified but are not recommended since they are more complex to apply and can lead to instabilities. Instead, it is recommended that the model be extended downstream to subcritical locations, and allow SRH-2D to compute the transition into and out of supercritical flow.

As is the case for a 1D model, a common value to specify for the downstream water surface at the boundary is a normal or critical depth. SMS has a tool known as the Populate tool (accessed using a button on the *SRH-2D Linear BC* dialog) that is used to compute and assign a normal or critical depth for the outflow boundary condition (see Figure A.17). A composite roughness value, slope, and flow are specified by the user when using the Populate tool. SMS extracts a ground elevation cross section from a specified underlying elevation data source (mesh or scatter set) that is used to compute the area/wetted perimeter. When using this option, care should be taken to be sure the BC arc is located within the boundaries of the elevation source specified (mesh or scatter). For this reason, it is recommended that the outflow BC arc be created just upstream of the most downstream part of the mesh.
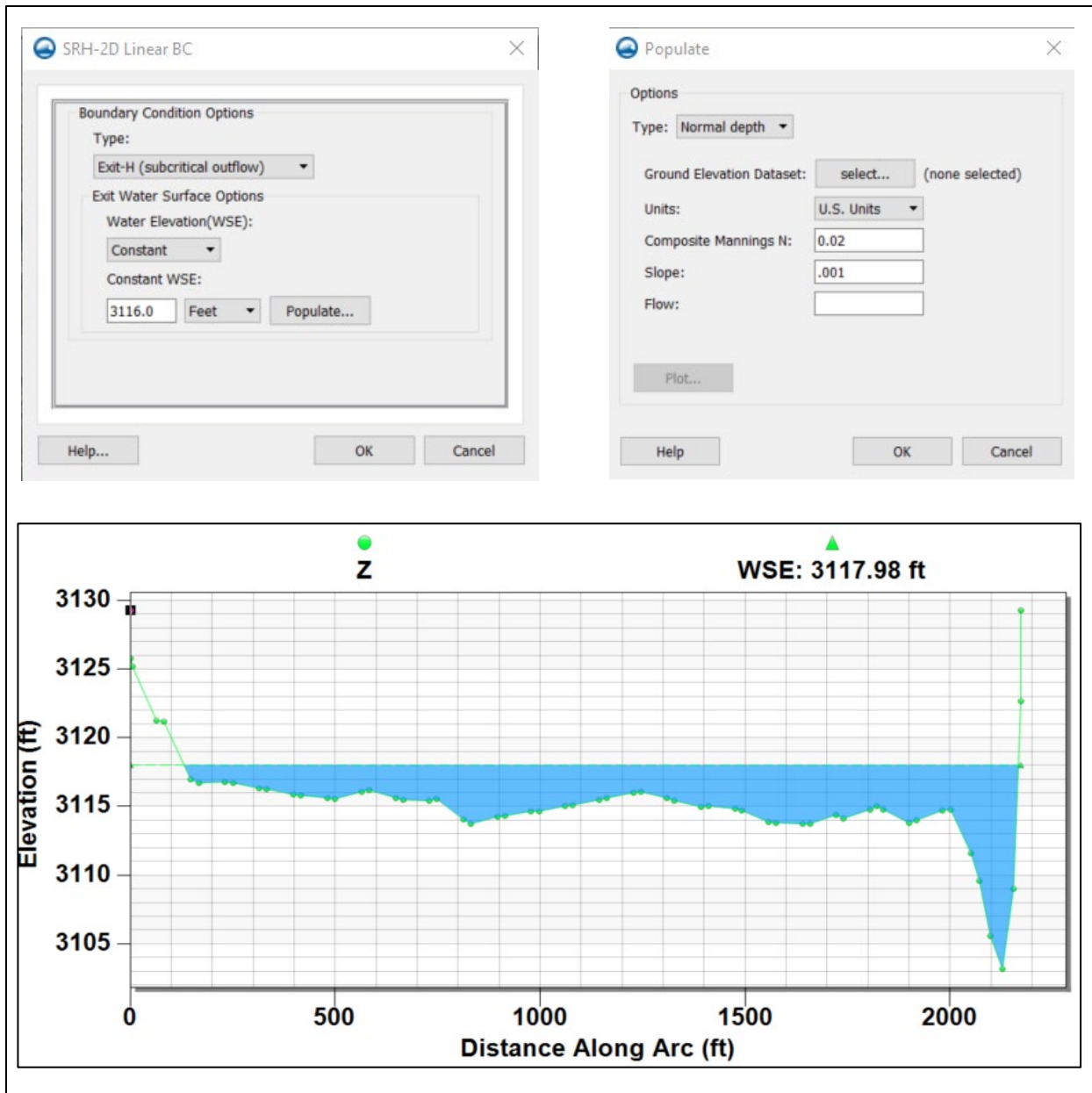
Figure A.17. SRH-2D dialog with normal depth calculation example.

Other BC types can also be specified. Closed or "no flow" boundaries are specified as either Wall (no-slip) boundary or a Symmetry (slip) boundary. Since no flow can cross these boundaries, they act as vertical walls. The Wall boundary type is the default applied to all boundaries of the mesh that are not assigned another boundary type. The Wall BC type is usually recommended, but the slip option may be used at smooth piers or other areas where there is a vertical wall with a smooth boundary; however, a Wall boundary provides similar results, especially if the elements adjacent to the wall are small. This is because the zero velocity at the element edge usually affects only one element. Another option is an Internal Sink that is used to remove (or add) flow inside the model domain.

A.4.4. Defining Materials

The next step is to define the materials for the model. This is done by creating a Materials coverage and specifying attributes to define the material zones and roughness values. The steps for creating material roughness zones in SMS for an SRH-2D model are detailed in the sections below.

A.4.4.1. Creating the Materials Coverage

Create a new coverage by right-clicking the Map Data folder in the *Project Explorer* and selecting *New Coverage*. Set the coverage type to *SRH-2D | Materials* in the *New Coverage* window. A new name can be specified if desired.

A.4.4.2. Creating Materials Polygons

Arcs are created in this coverage and connected to form polygons representing surface roughness zones for the model (see Figure A.18). Material polygons are generally digitized by hand from an aerial image but can also be imported from a shapefile.

Once arcs defining the material polygons are created, be sure to execute the *Feature Objects |* Build Polygons command to create the polygons. It is best to generate material polygons that extend beyond the model domain extents to be sure all mesh elements are within a polygon. It is helpful to display the mesh boundary when creating these polygons. Materials are assigned to each mesh element based on the polygon containing the centroid of the mesh element.
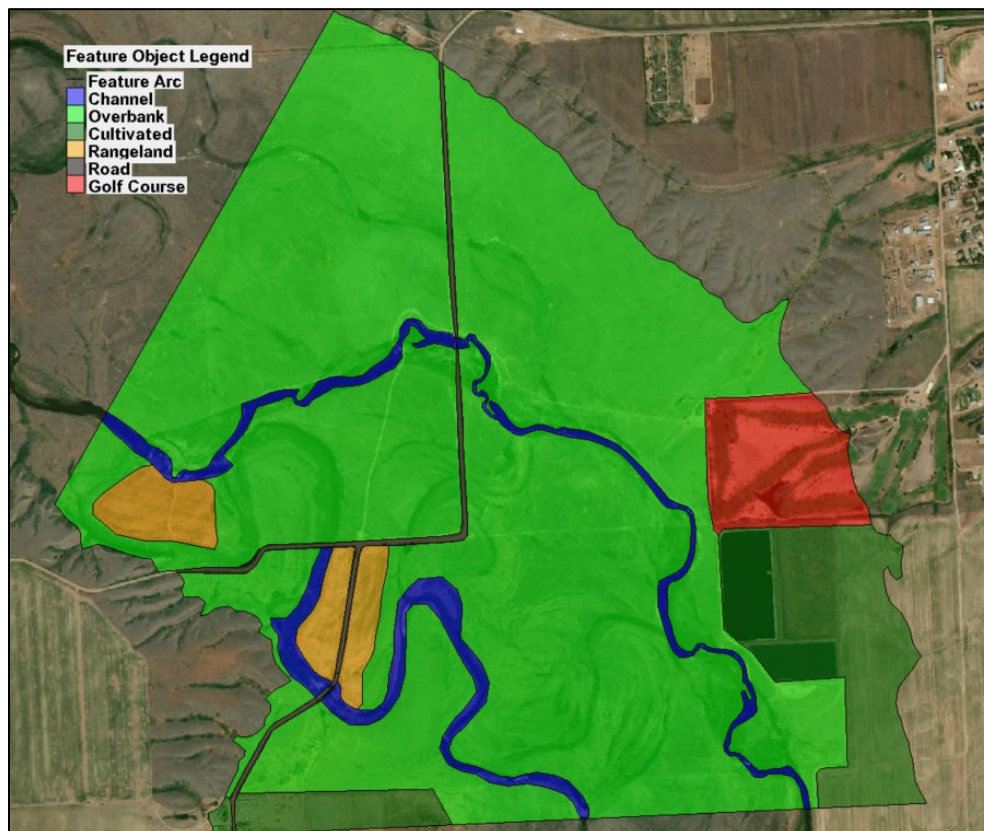

Figure A.18. 2D materials color fill display.

A.27

A.4.4.3. Specifying Material Types

The next step is to define a list of materials to be used for the model. Materials should be chosen based on information in section 4.4.1. This is done by right-clicking on the Materials coverage and selecting Material Properties. In this dialog (see Figure A.19), a list of materials is created. When first opened, the only material in the list is "unassigned." This material designation is always available; it is the default material and is used to specify areas of the mesh that cannot convey water (similar to a hole in the mesh). Each material can be assigned to multiple polygons.

Materials are added to the list by selecting the [+] button. A material type can be renamed by selecting the name text to change it to the desired name. The [−] button is used to delete a material. The buttons to the right of the material names are selected to change the color or pattern for the display of the material polygons. A Manning's roughness is also specified for each material. Roughness values are specified as constant or depth varying.
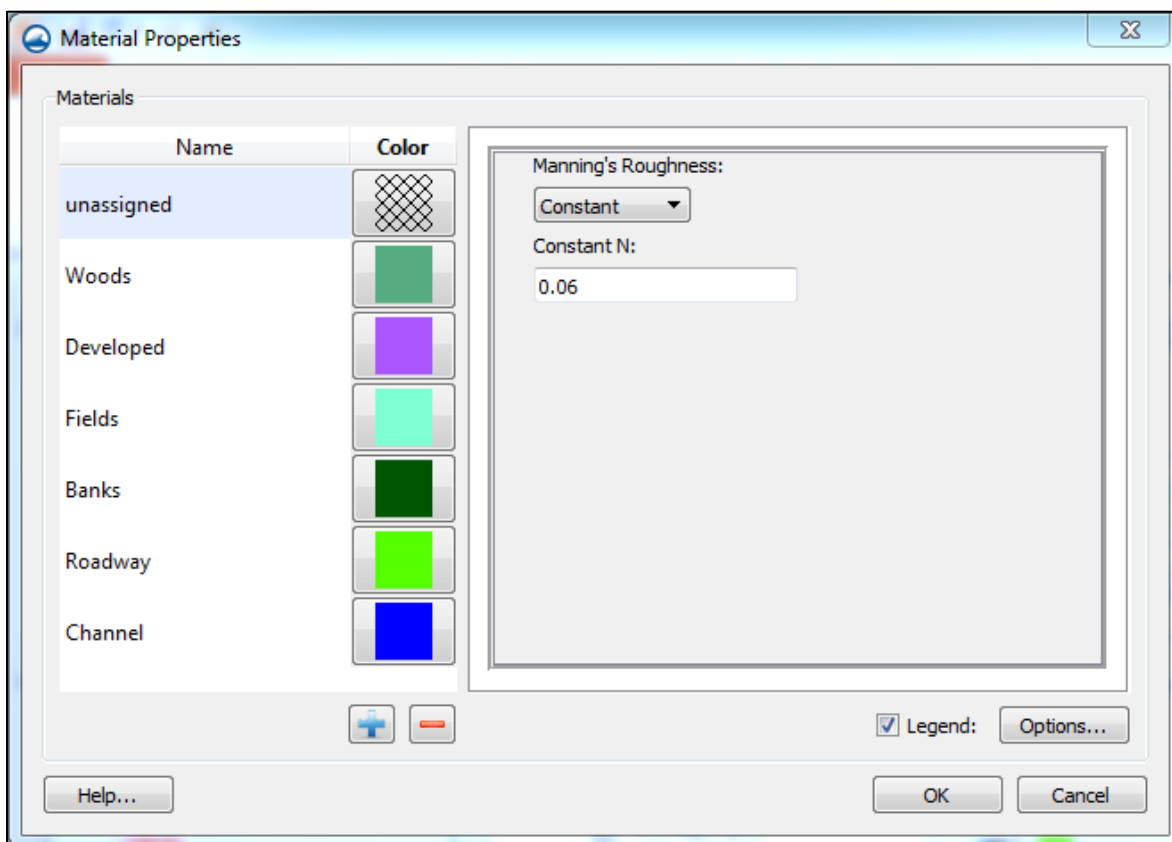

Figure A.19. 2D materials properties dialog.

A.4.5. Assigning Materials to Polygons

Once the material list is created, each polygon in the Materials coverage should be assigned a material from the list. This is done by choosing the Select Feature Polygon tool [ ] and double-clicking on each polygon. This opens the *Material Properties* dialog again. Selecting a material name highlights that material in the table and define it as the material specified for the selected polygon. Once a material is assigned for all polygons, use the *Polygon Fill* option in the *Map* section of the Display Options dialog to review material assignments.

A.4.6. Create Monitoring Points and Lines

The next step is to create a Monitor coverage for the model. While optional, it is recommended that a monitoring coverage be included in every SRH-2D simulation. The Monitor coverage contains points and lines where data is extracted from the model solution. Water level values for Monitor points are displayed during the model run to help monitor the model progress and determine model equilibrium. Monitor Lines are used to extract flows from the model and are used to verify model continuity and stability. The steps for creating monitor points and monitor lines in SMS for an SRH-2D model are detailed in the sections below.

A.4.6.1. Creating the Monitor Coverage

Create a new SRH-2D Monitor coverage by right-clicking on the Map Data folder in the *Project Explorer* and selecting *New Coverage*. Set the coverage type to *SRH-2D | Monitor*. A new name can be specified if desired.

A.4.6.2. Defining Monitoring Points

Create points in the coverage using the Create Feature Point tool ⁘ at key locations where data should be evaluated in the simulation. Usually, one is created near the outflow boundary and another near the inflow or perhaps near an area of interest. Water levels for all monitor points created can be plotted during the model run.

Monitor points should be created at other locations where it is desirable to extract values from the solution. SRH-2D extracts values from the solution datasets at the monitoring point locations. This data is written to a separate output file in the output directory called PT(n).dat, where n is a sequential integer corresponding to the order of the SMS feature point ID numbers for the monitor points.

A.4.6.3. Defining Monitor Lines

Monitor lines are created to extract data from the solution along an arc. Monitor lines are optional but highly recommended to help verify continuity and model stability. Create monitor lines using the Create Feature Arc tool and digitizing arcs. SRH-2D calculates the total flow across the arc and the average water surface elevation along the arc. Create the arcs in the Monitor coverage at locations where it is desirable to compute flow. The direction of the monitor line does not matter, except that flow values are reported as positive or negative if the line creation direction is reversed. Flow values for all monitor lines created can be plotted during the model run. It is recommended that monitor lines be created near the upstream and downstream boundaries. Monitoring flows near the outflow boundary can help verify model continuity. Monitor lines are also useful for evaluating flow splits or other areas where the flow across a designated line is desirable.

Monitor line output is written to a separate output file in the output directory called LN(n).dat, where n is a sequential integer corresponding to the order of the SMS feature arc ID numbers. Monitor line output can help determine when steady flow is achieved since the total flow at a monitor line near the outflow boundary should equal the total inflow at steady state. Once steady state is achieved, flows reported in these files corresponding to monitor lines placed near the outflow should be less than about 2% of the total inflow. Also, reviewing the flow and WSE values reported in these files at the final time steps can help verify model stability. Unstable models may show varying or oscillating values for the flows reported for the monitor line (see Figure A.20).

Figure A.20. Monitor line output file.

A.4.7. Model Simulation Setup

Once the mesh, Boundary Conditions coverage, Materials coverage, and Monitor coverage have been created, the next step is to create an SRH-2D simulation and link these four components to the simulation. The steps for creating a simulation in SMS for an SRH-2D model are detailed in the sections below.

A.4.7.1. Creating a Simulation

SMS uses a simulation-based approach to modeling. A Simulation in SMS is a collection of components that define the model. SMS organizes data for the SRH-2D model runs into one or more simulations created in the *Project Explorer*.

Simulations need three components to run: a Mesh, a Boundary Conditions coverage, and a Materials coverage. It is highly recommended that a fourth component, a Monitor coverage, also be included in every simulation. Model Control parameters should also be defined for each simulation.

Create a simulation by right-clicking in the blank part at the bottom of the *Project Explorer* and selecting *New Simulation -> SRH-2D* (see Figure A.21). The first time this command is used, a hierarchy of folders is created that are used for SRH-2D simulation components. SMS allows for the creation of multiple simulations. It is often helpful to create a new simulation by duplicating and modifying an existing simulation.
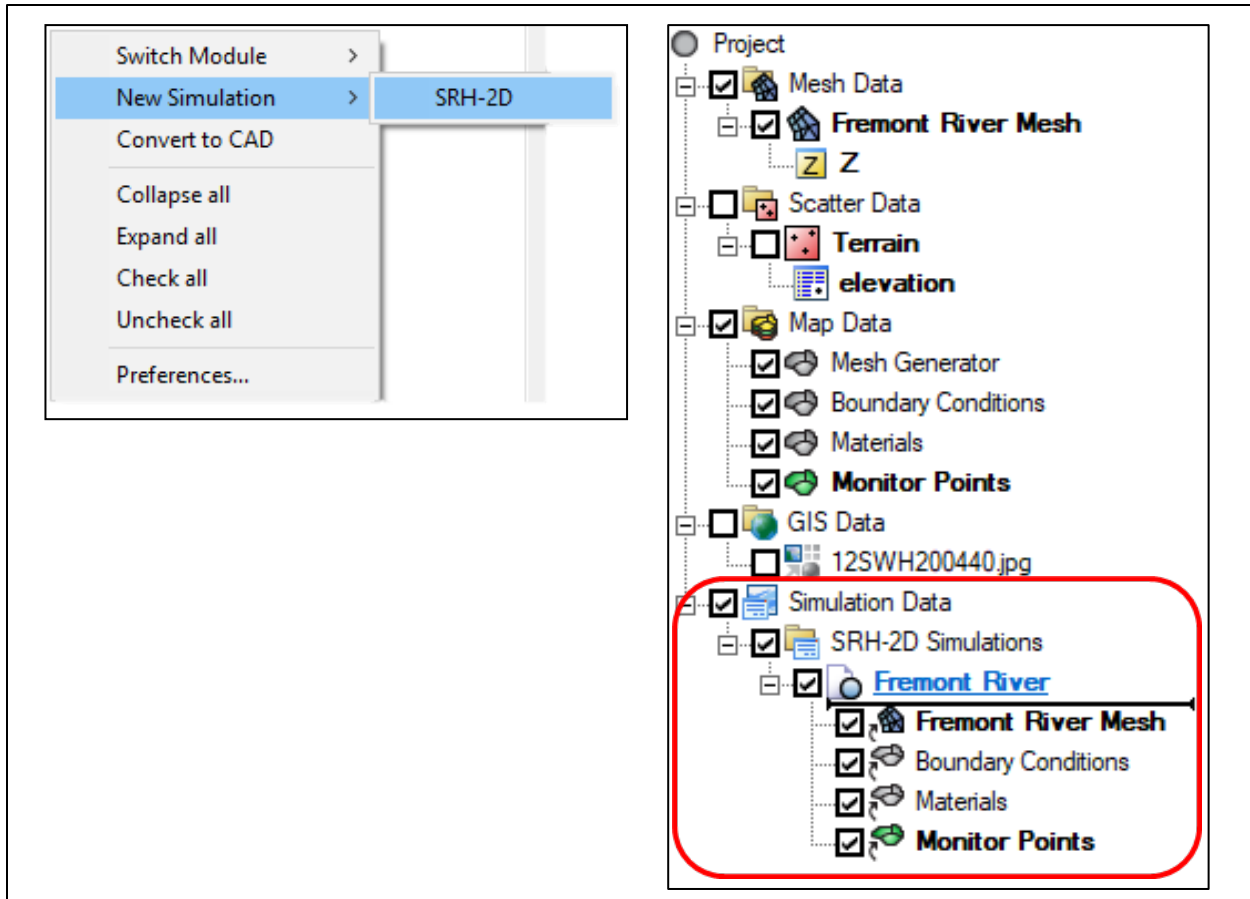
Figure A.21. Simulations in the project explorer.

A.4.7.2. Linking Simulation Components

Once a simulation is created, the four components should be "linked" to the simulation. The link references the source of the data. The use of links allows items to be shared between multiple simulations without duplicating data. Links are created by clicking on the component in the *Project Explorer* then dragging and dropping the component just below the simulation name. A dark underline appears below the simulation, indicating it should be dropped. Components can also be linked by right-clicking the component and selecting *Link to | SRH-2D Simulations -> <simulation name>*.

Generally, each SRH-2D simulation has the following components linked to it:

- Mesh
- Boundary Conditions coverage
- Materials coverage
- Monitor coverage

An Obstruction coverage can also be included but only if modeling structures as obstructions. More details on using an Obstruction coverage to define structures are included in section A.5.7. of this appendix.

A.31

A.4.7.3. Snap Preview

Model parameters such as boundary conditions and material properties are assigned to feature objects defining the location where the parameters are applied to the model. However, for SRH-2D these locations should correspond to element edges within the mesh. As part of the process of mapping the coverages to the mesh parameters specified for BC arcs or arcs defining polygon boundaries are "snapped" to the nearest mesh element edges. Snapping is not done until a model simulation exists and both the coverages with the feature objects and the mesh are linked to the simulation.

The element edges that have the parameters from linked feature objects can be viewed by turning on the "Snap Preview" in the Display Options dialog under the "Map" section. This option is only available once the coverages and a mesh are linked to a simulation. Turning on the "Snap Preview" display option in SMS shows a preview of how the feature objects are "snapped" to the element edges using dotted lines to represent the snapped element edges. Figure A.22 shows the location of the snap preview option in the *Display Options* dialog. Figure A.23 shows how the snap preview lines appear in the main view window.
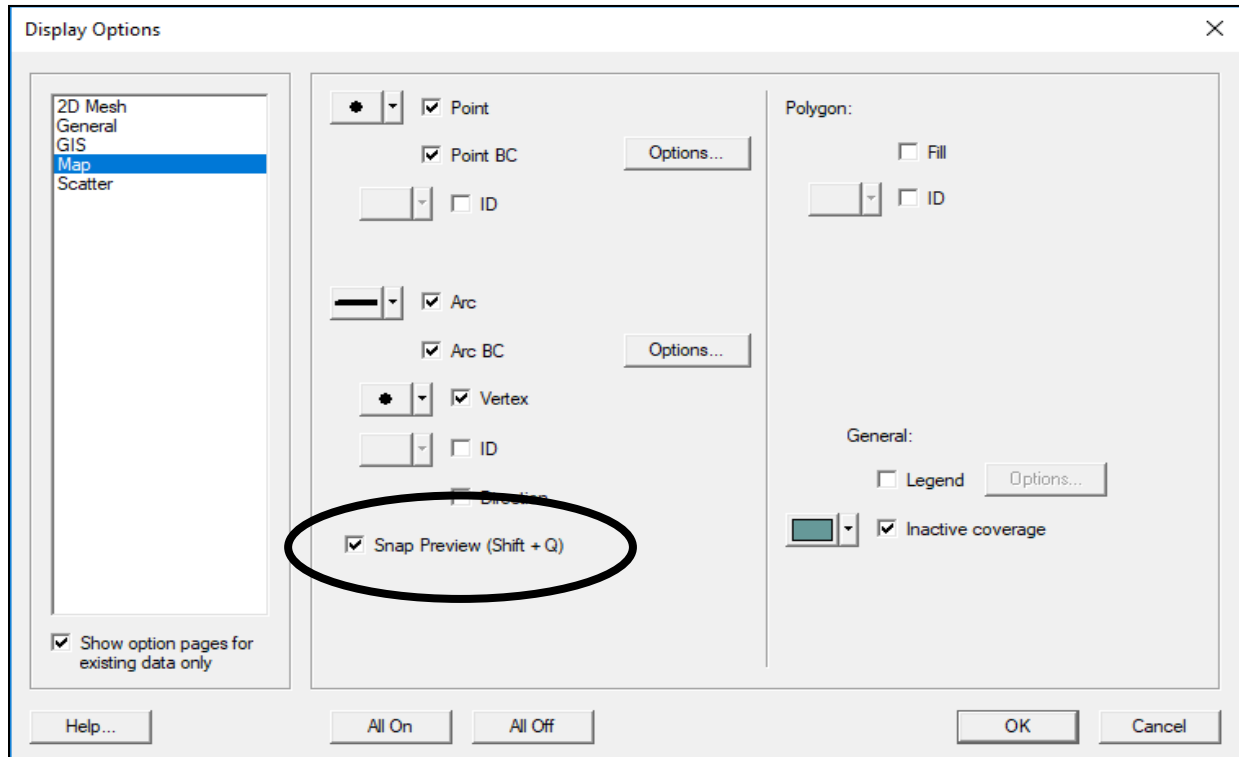

Figure A.22. Display options dialog box showing the snap preview checkbox location.
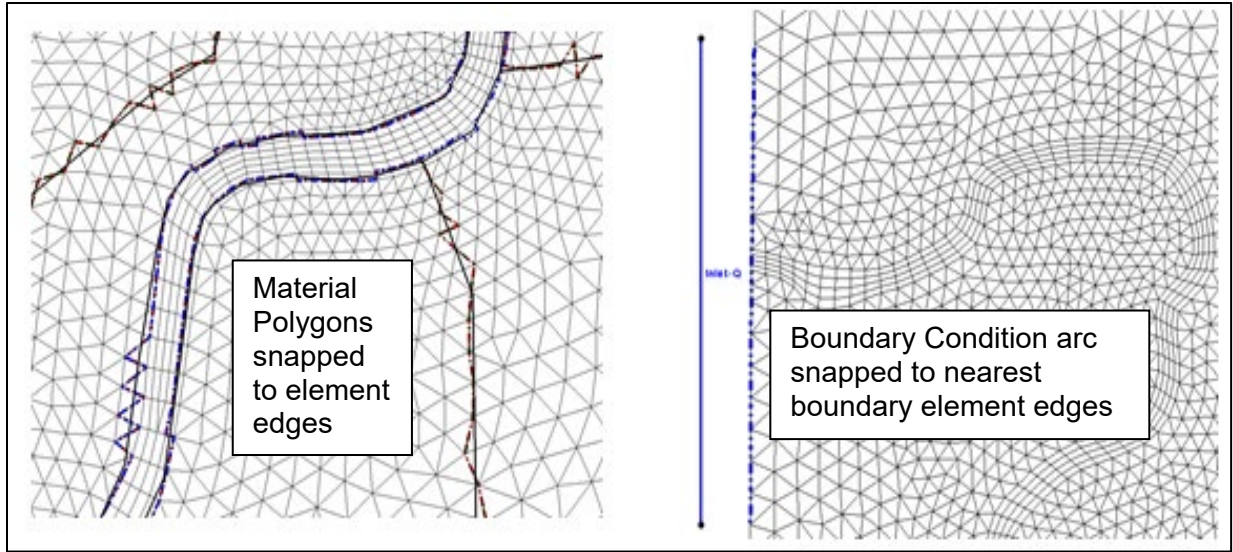
Figure A.23. Snap preview display.

A.4.7.4. Setting Model Controls

The last step in defining an SRH-2D simulation is to set the Model Control parameters to define the options for running SRH-2D. To set model control parameters, right-click on the simulation, and select Model Control.

The parameters are divided into three tabs.

General tab

- **Simulation description** - This is simply a text description written to the output files for documentation

- **Case name** - A one-word identifier used during SRH-2D execution and used as a prefix for all output files written to the output directory

- **Time controls** - *Start time*, *end time,* and *time step*. The start time is usually set to 0.0. The end time defines the duration of the model run and varies depending on the type of model. For a steady-state model that starts dry, an end time is selected such that the model has time for the water to flow through the domain, fill all storage areas, and reach a point of equilibrium. If the model starts partially wet, then this time may be less. Time for a transient run with a hydrograph should allow time for base flow to be established and for the hydrograph to propagate through the reach and return to base flow conditions. The time is often estimated, then reviewed after the model runs and adjusted as needed.

- The time step is quasi-dependent on the Courant Condition that is dependent on the mesh resolution and water velocity. This generally means that water should not flow more than one element length during a single time step. Since SRH-2D uses an implicit scheme, it is not strictly governed by the Courant criteria. These criteria can be used to estimate a time step, however, in practice, a model is usually run with a time step of 1 to 5 seconds as a starting point. If the simulation "crashes," indicating that the solution diverged, reduce the

A.33

time step and try again. Time steps smaller than 1 second are not typically needed, if so, it may indicate other problems in your model.

- **Initial conditions** - This sets the initial conditions at the first-time step of the model run. For most models, the initial condition is specified as dry. Descriptions of the four available options are provided below:

  1. *Initial Water Surface Elevation* - This specifies that still water is present in all areas below a specified elevation. This would be used if there is a lake or other large storage area that may take a long time to fill, or it can be used to fill a portion of the channel or floodplain.

  2. *Dry* - This is usually the recommended option. It assumes that the model is dry and water flows into the model at the first-time step.

  3. *Automatic* - This option specifies an initial WSE equivalent to the downstream WSE BC setting and is similar to the first option. It should be the method chosen when using a time-varying (tidal) outflow boundary, as the Dry initial conditions option does not allow any water to flow upstream into the model until water begins to flow out of the model. Also, if a rating curve is specified, this option begins the simulation using an elevation corresponding to the lowest point on the curve.

  4. *Restart* - For this option a solution from a previous model run is used, sometimes referred to as a "hot start" for other models. This is used to start from a given flow condition such as a base flow condition and then run the model from that point. It can also be used to start from an intermediate point for a model that failed. This can save time by not having to begin the model from the start. This option only works for models with the same mesh. If the mesh is altered in any way, this option cannot be used. Restart files are found in the output directory, one for each output time interval is written and called RST(n).dat, where n is the output interval index.

Flow tab

The *Turbulence Model* is the only option set under this tab. The default is the *Parabolic* turbulence model. For this model, the flow resistance due to turbulence is dependent on the velocity gradient. This model works well in SRH-2D, and it is recommended that this option always be selected. The *Parabolic Turbulence* parameter is a coefficient applied to the turbulence force. Generally, this is also left at the default value of 0.7 but can vary between 0.1 and 1.0 and be used as a fine-tuning parameter for calibration. One specific case is when SRH-2D is used to support flume modeling. For most laboratory flume conditions, a turbulence coefficient of approximately 0.2 would likely be more representative.

Output tab

Under this tab, the output format, units, and frequency are specified. The output format is always the default option of XMDF if using SMS to review results. The output units are set to either English or SI and can be different from the units used to build the model.

The model computes a solution at every time step, but to save on disk space, the solution is usually written at a subset of times as specified by the user in the *Result Output Method* section. There are three options for *Result Output Method*. The most common is to specify an output frequency. This sets the time interval between output solutions. The other options are to specify times to write output or to specify output only at the end of the final time step.

A.4.8. Running the Model

A.4.8.1. Model Checks

SMS includes a model checker to test for common issues in running a model, such as leaving out a required component. The model checker automatically runs when launching a model. The model check can also be run independently. To do this, right-click on the simulation and select *Model Check*. This reviews the model to identify problems that may result in errors or termination of the model. Keep in mind this function does not catch all potential errors in setting up a model but can help identify some common issues.

A.4.8.2. Running the Model

Once the model simulation is set up, the model is ready to run. This is done by right-clicking on the simulation and selecting the *Save, Export, and Launch SRH-2D* command. This command saves the SMS project, exports the model input files needed to run SRH-2D and finally launches SRH-2D, which reads the input files and runs the model. Each of these functions can be executed independently, but most often are done all at once using this command.

SRH-2D is comprised of two separate programs. The SRH-2D preprocessor (SRH-Pre), processes the exported data files and creates a simulation file. SRH-2D runs the simulation, using the simulation file created by SRH-Pre.

The *Save* and *Export* commands write a series of input files. These include both SMS project files and SRH-2D input files. The SMS project files are saved in a directory specified by the user. The SRH-2D input files are saved in a series of subfolders found in the directory where the SMS project is saved. The subfolders are named with the SMS project name, followed by an "SRH-2D" folder, and finally, a folder carrying the name of the simulation (see Figure A.24). The prefix of the SRH-2D input files is the project name.
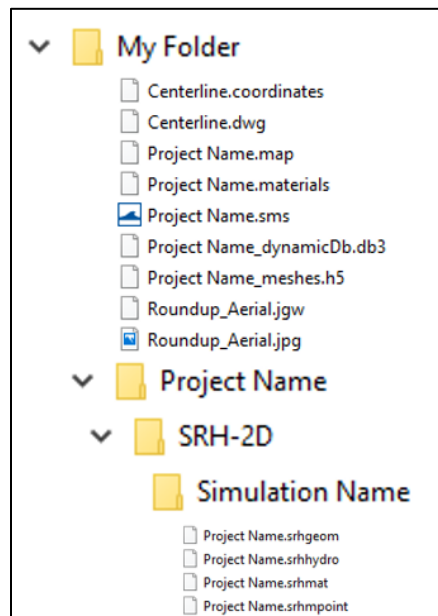

Figure A.24. Snap preview display.

A.4.8.3. Simulation Run Queue

When SRH-2D is launched, SMS opens and display the *Simulation Run Queue*. This tool displays the status of the model run. Multiple simulations can be run simultaneously. The *Maximum Number of Concurrent Processes Allowed* setting manages how many simulations can run at once. If more simulations than the specified number exist in the queue, the additional simulations remain on hold until the others finish, freeing up a process and allowing a new one to start running. If a simulation is launched, that same simulation cannot be launched again until it has completed. When additional simulations are launched, they are added to the *Simulation Run Queue* (see Figure A.). For each simulation, the model status output for either SRH-Pre or SRH can be displayed, as well as various plots—such as a water level plot at the monitor points.

Select the drop-down arrow to the left of the simulation to expand the simulation and display the SRH-Pre and SRH-2D run status during the run. Click on either of these two lines to display any executable run messages and text in the lower portion of the dashboard. This is where application errors are reported.

If the simulation run queue is closed, it can be accessed again by right-clicking on *Simulation Data, SRH-2D Simulations*, or the simulation itself in the *Project Explorer* and selecting *Simulation Run Queue….*
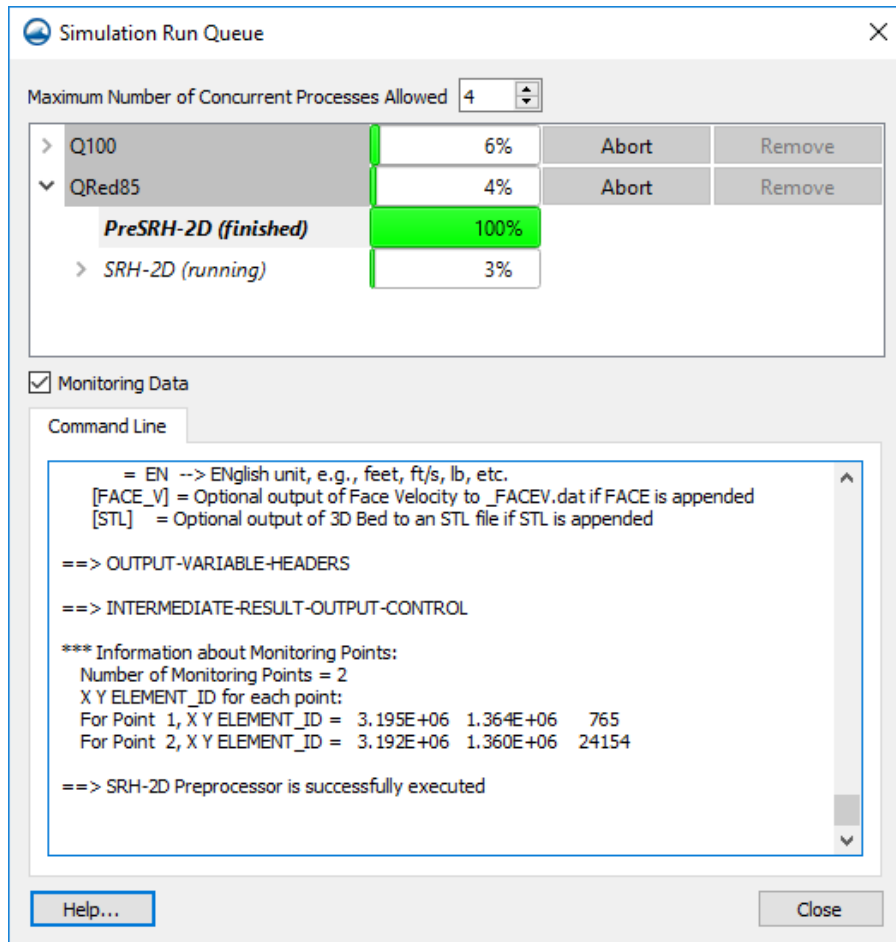

Figure A.24. Simulation run queue.

A.4.8.4. Error Messages

Error messages can be present in either the lower portion of the dashboard or in the SRH output summary file. Expand the simulation and click on either *PreSRH-2D* or *SRH-2D* for the simulation to check for any errors that may have been reported. Messages for common SRH-2D Errors are documented in the SMS wiki, including a description of the error and a potential solution to correct the error.

A.4.8.5. Plot Options

Several different plots can be displayed during a simulation run, including water surface elevations at the monitoring points and computed flow along the monitor lines. The plots should be turned on before viewing them. To turn on the various plots, expand the SRH-2D section of the simulation to expose the *Plot Options* button. The display of the plot can be modified with various line options. A *Progress Update Interval* can be set specifying how frequent the plots are updated. Figure A.25 shows a plot displaying the computed flow along the monitor lines and the plot line options.
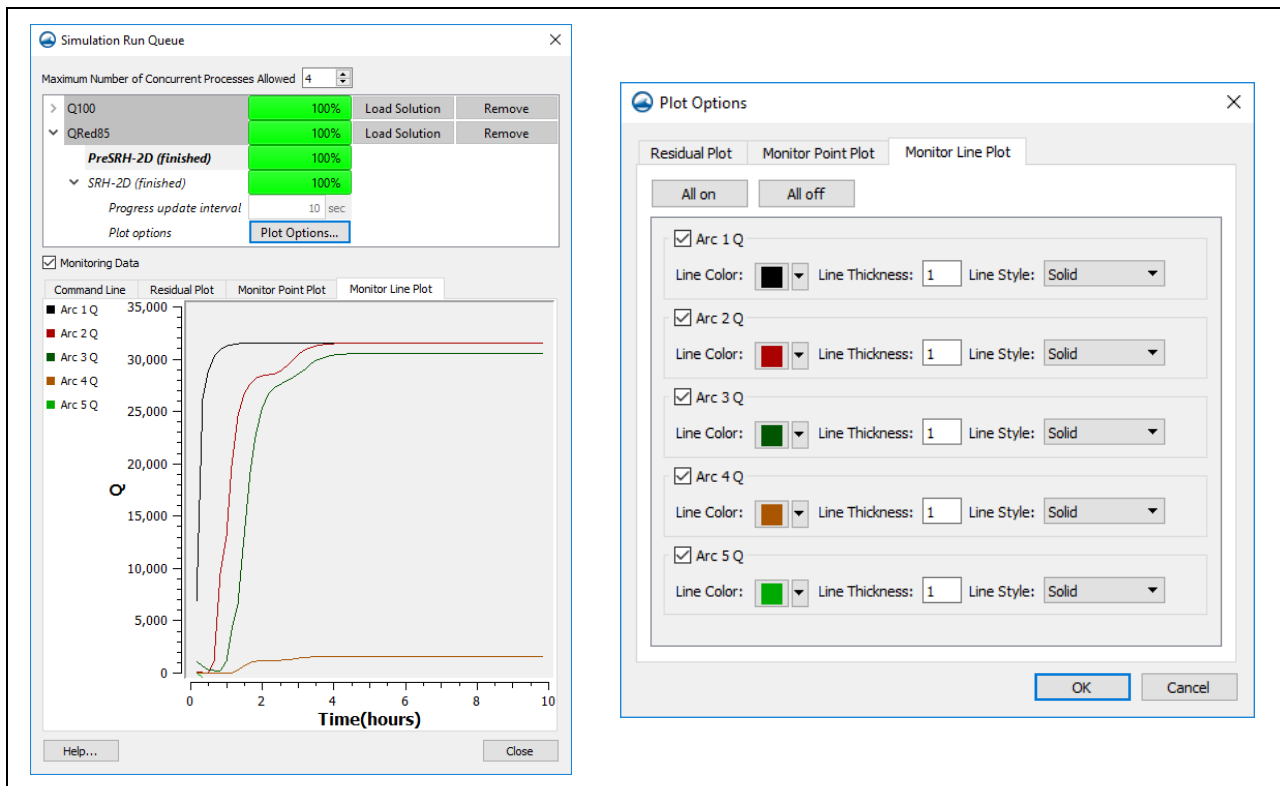
Figure A.25. Simulation plots.

A.4.8.6. Model Output

Once the model runs successfully, the *Abort* button changes to a *Load Solution* button. Click the *Load Solution* button to load into SMS the set of solution files that were written by SRH-2D during the model run. The mesh datasets are organized in a folder carrying the name of the simulation (see Figure A.26). The solution files are saved in the same subfolder where the SRH-2D input files are located. The prefix of the output files is the case name specified in the simulation *Model Control*.
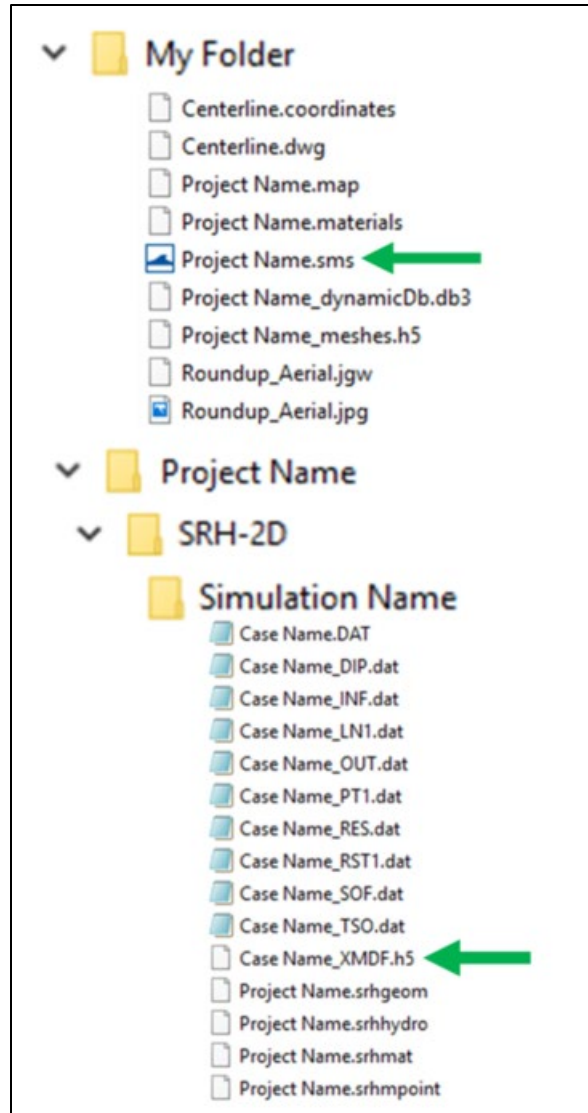
Figure A.26. SRH-2D output files.

A.5. MODELING BRIDGES AND OTHER STRUCTURES

Structures are represented in SRH-2D either directly in the 2D domain or by adding them as 1D objects (see Figure A.27). It is generally recommended to represent structures in 2D when feasible. A 2D structure is represented by modifying the mesh elements and elevations to capture the geometry of key features of the structure.

One-dimensional structures create flow connections between different sections of the 2D mesh. Flow leaves the 2D mesh at the upstream BC arc and re-enters the mesh at the downstream BC arc. SRH-2D disables the 2D elements between the BC arcs. Flow through the structure is defined by 1D equations for the structure. The figure and lists below show the 1D and 2D applications for defining structures in SRH-2D.

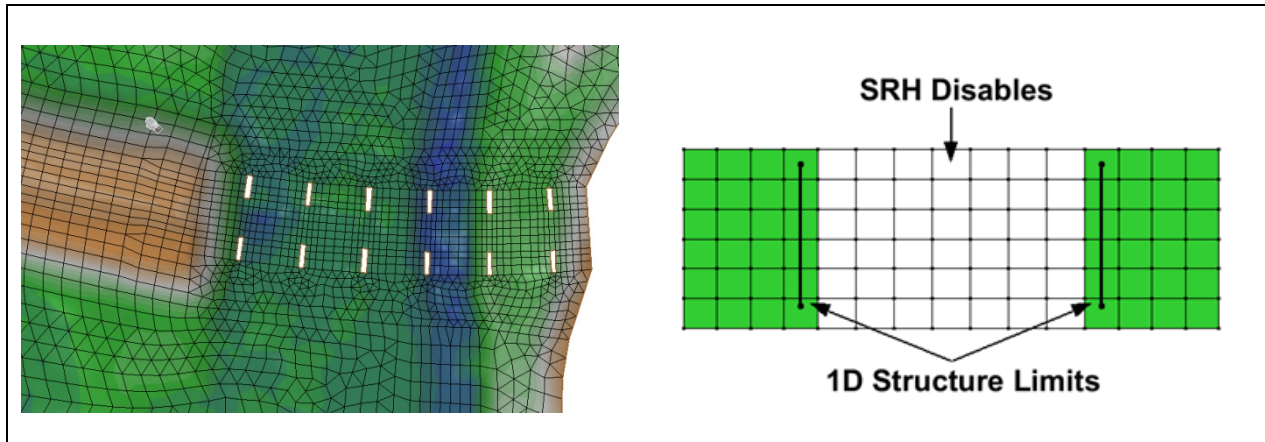| | | |
In Mesh (2D domain)		1D Representation
- Embankments		- Culverts
- Abutments		- Weirs or Overtopped Bridges
- Guidebanks		- Gates
- Piers		- Diversions (Links or Internal Sinks)
- Box culverts		

Figure A.27. 2D and 1D structures in SRH-2D.

The following sections detail how and when to represent various structures using the 1D and 2D approaches with SRH-2D using SMS.

A.5.1. Representing Bridge Piers

Piers are represented explicitly as holes in the mesh or as a flow obstruction (see Figure A.28).

Piers as holes in mesh   Piers as obstructions
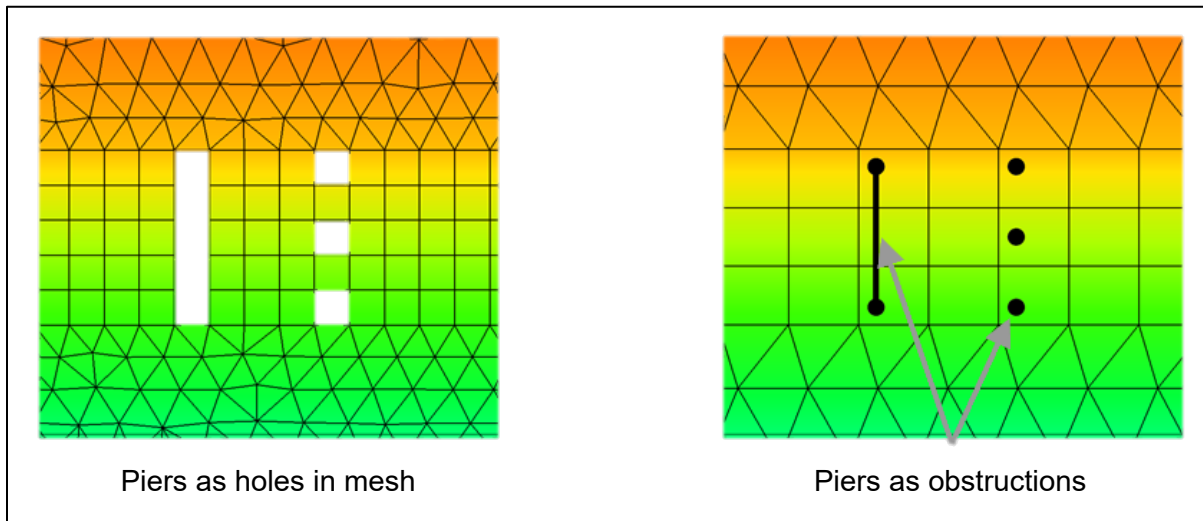
Figure A.28. Representing bridge piers.

Bridge piers are most often represented by creating "holes" in the mesh that represent the shape of the piers (as shown in the figure above on the left side). This forces the water to flow around the piers and accounts for the resulting head loss. Boundaries at holes in the SRH-2D mesh represent vertical walls, with a zero velocity at the face. A roughness value is not associated with

A.39

the pier wall. Additional shear stress may be added as a boundary condition along the pier face to represent additional head loss, but this is typically only used if the pier loss is significant.

Including piers directly in the mesh has the advantage of more accurately representing the location of the pier and provides better visualization of the flow paths, as well as more representative velocity and shear stress distributions.

To create holes in a mesh, create a separate polygon for each pier. The polygons are assigned a mesh type of "None" so no elements are created within this polygon. Generally, the pier width would be used for the element size. It is a reasonable simplification to represent circular piers as square elements in the mesh. For larger piers, the element size may be less than the pier width. It is recommended that the piers not be modeled at very high resolution to avoid unnecessary complexity in the model. Figure A.2630. shows various ways piers may be modeled as holes in the mesh based on the guidelines above.

Guidelines for representing piers as holes in the mesh include:

- The pier footprint (polygon) should represent the average pier dimensions over the length of the pier stem that is below the WSE

- Pile caps may be represented in the terrain data

- Additional pier width may be added to account for debris accumulation

- Recommended element size at a pier:
  1. Minimum = Pier width/3
  2. Maximum = Pier width

- If the element size is less than the flow depth, instabilities may form. Therefore, it is recommended that element size be greater than the flow depth. If using element sizes less than the flow depth, the stability of the solution in these areas should be monitored.
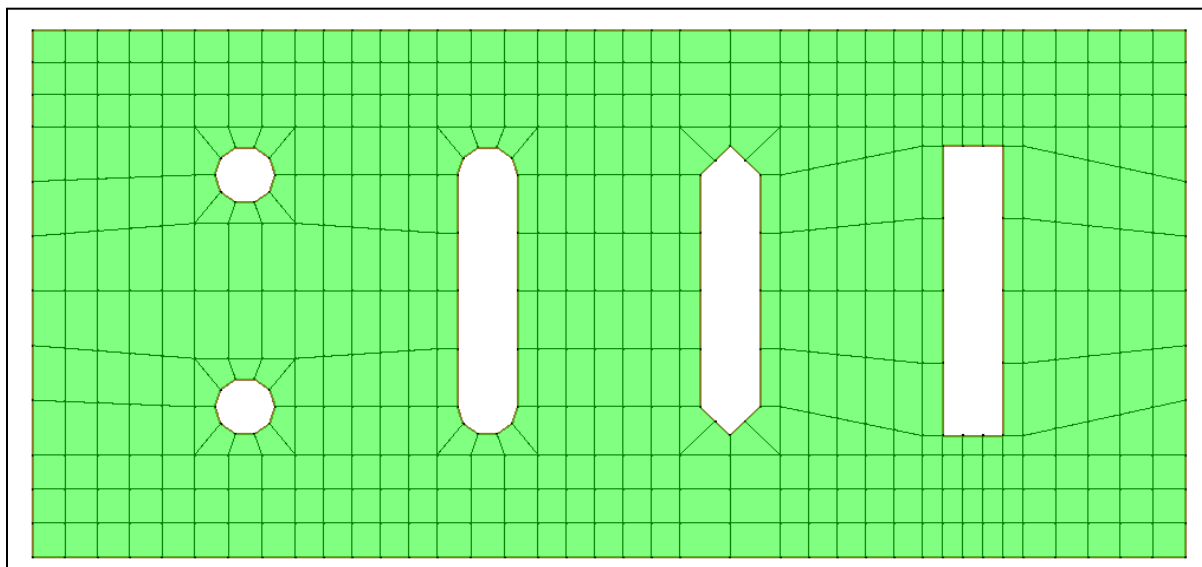

Figure A.29. Modeling piers as holes in the mesh.

Piers can also be represented in SRH-2D as "flow obstructions" (as shown on the right side Figure A.28.). Flow obstructions can adequately account for the head loss due to the piers, but not the detailed flow characteristics around the piers. Therefore, it is usually recommended to model piers as obstructions only for secondary bridge locations that are not the main area of focus of the model. Flow obstructions are assigned to an X, Y location with geometric dimensions and result in an additional drag force applied to the given element based on the pier geometry. Flow obstructions are a simple approach that apply an additional drag force and do not involve any modification to the mesh. More detail on creating obstructions using SMS can be found in section A.5.7 of this document.

Using holes in the mesh is the preferred approach if a detailed representation of velocities and flow direction through a bridge opening is needed. If not, the use of obstructions represents the average head loss associated with the piers and results in the same water surface elevation upstream of the bridge, when the appropriate drag coefficient is used.

A.5.2. Bridge Pressure Flow

The preferred method for representing pressure flow at bridge decks is to directly represent the bridge deck in the mesh and use pressure flow arcs in the boundary conditions coverage to define the pressure zone. Note that it is not required to represent the bridge deck in the model if the water surface does not go above the low chord for the flow conditions being modeled. However, including a pressure flow bridge in the model does not cause any problems as it is ignored if the water surface elevation at the bridge never reaches the ceiling elevation (low chord).

Pressure zones are defined within SMS by creating two arcs representing the upstream and downstream faces of the bridge deck in an SRH-2D Boundary Conditions (BC) coverage. The area between these arcs defines the region of the mesh corresponding to the pressure zone. These arcs are created in the same BC coverage that contains the inflow and outflow boundary conditions. When creating the arcs, be sure they are drawn in the same direction. It is best practice to use quadrilateral mesh elements aligned with the bridge deck between pressure zone arcs (see Figure A.30).
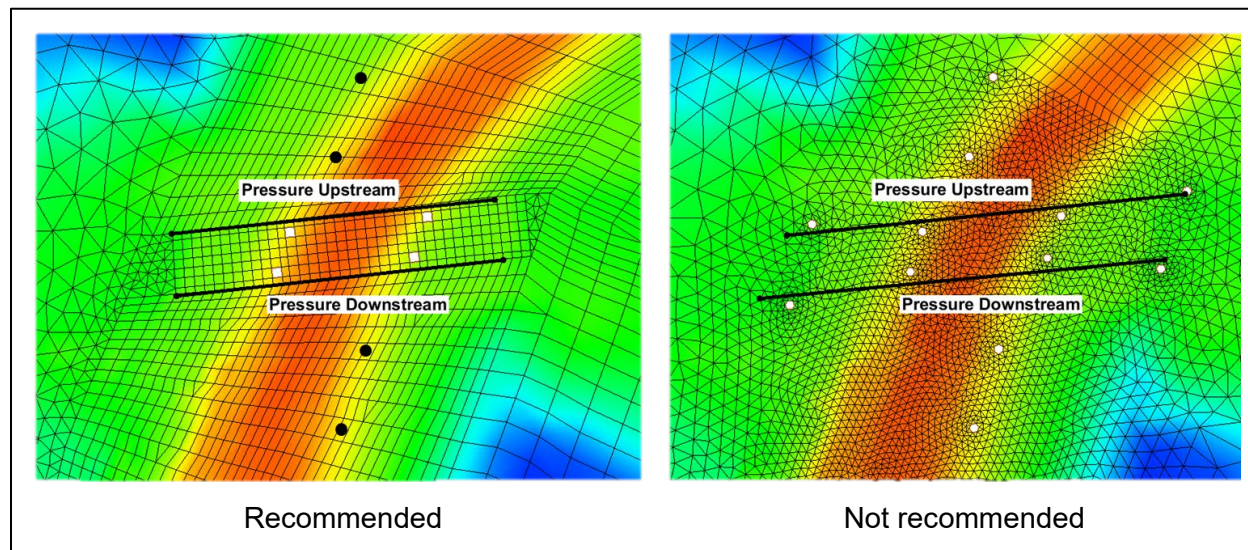

Figure A.30. Modeling pressure zones.

To define a pressure flow zone, select both arcs defining the upstream and downstream ends of the pressure zone then right-click one of the arcs and select *Assign Linear BC*. This launches the *SRH-2D Linear BC* dialog. The *Pressure* option is selected, and additional attributes can be assigned. Attributes to define the pressure zone include ceiling type (flat or arched), ceiling elevation for both the upstream and downstream faces, and a Manning's roughness for the ceiling. For flat bridges, there is a single elevation value in the direction of the roadway. Bridges with slopes in the direction of the roadway can be represented by creating multiple pairs of boundary arcs at different elevations.

The Manning's roughness coefficient for the underside of the bridge is currently not well defined, but model results are generally insensitive to this parameter. There is current research being conducted to provide guidance information for specifying roughness for bridge decks.

If needed, overtopping of the bridge deck can be specified in this dialog. Overtopping is modeled in SRH-2D as a 1D representation of flow between the pressure flow arcs only. The top of the roadway, or the top of the guard rail, is entered as the crest for overtopping. The length is the lateral weir length along the bridge deck where overtopping would be expected. The remainder of the overtopping flows over the embankment areas outside the arcs representing the bridge deck and is represented in the 2D flow.

A.5.3. 1D Structures

Hydraulic structures are represented in SRH-2D by coupling 1D computational solutions for common structures with the 2D model. This is done by creating two arcs in the SRH-2D Boundary Conditions (BC) coverage to represent where the flow exits (upstream arc) and then re-enters (downstream arc) the 2D model domain. The actual boundary for these structures is "snapped" to the element edges nearest to the BC arcs for each structure. Elements between arcs representing extents of the hydraulic structure are "disabled" by SRH-2D and SRH-2D does not convey flow through these elements (see Figure A.31). Attributes should be assigned to the BC arcs to define the 1D structure characteristics.
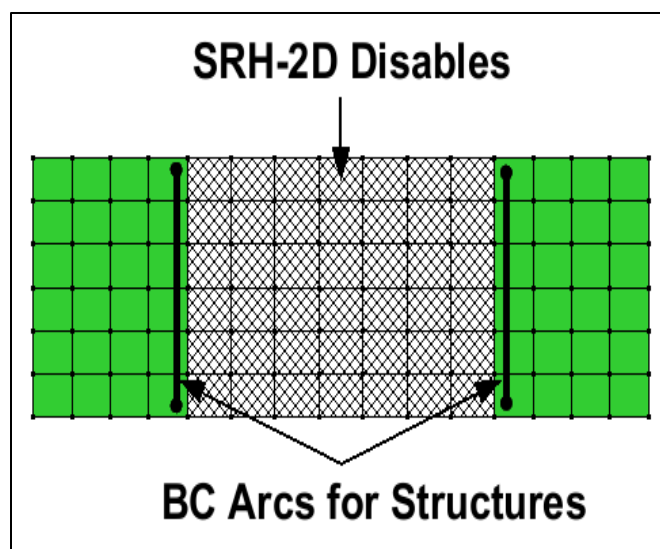

Figure A.31. 1D structures in SRH-2D.

At each time step in the solution, the flow for a 1D solution is computed based on the WSE at the structure boundary. This flow is then applied at the boundary for the next computational time step for the 2D solution.

A.5.4. Culverts

A.5.4.1. 1D Culverts

Culverts are typically represented in SRH-2D models as 1D structures. The HY-8 culvert hydraulic analysis program is used to compute the solution for 1D culverts in SRH-2D. Since the actual HY-8 computational engine is coupled with SRH-2D, any culvert that can be represented in HY-8 can be modeled in conjunction with SRH-2D. Multiple culverts can be defined for a project, but all should be defined in the same HY-8 file.

As with all 1D structures represented in an SRH-2D model, the structure is defined in the BC coverage using two arcs. When creating arcs, be sure to draw the arcs in the same direction (arc direction can be verifying by turning on arc direction in the Map module Display Options). These arcs should be placed at the approximate location where the culvert flow exits the 2D mesh to flow into the culvert and the location where the 1D culvert flow returns to the 2D mesh. Using the widest distance between the culvert wing walls as the arc length is a reasonable estimate in most cases. Care should be taken to be sure the elevation of the element edges corresponding to the culvert boundary arcs are at an elevation equal to the culvert invert elevations. If the 2D mesh elevations at the boundary are too high it can impede flow into the culvert or cause instabilities. Finally, the mesh in the area between the culvert BC arcs should be constructed using near orthogonal quadrilateral elements if possible. This helps better define how elements between the arcs are disabled and results in better computational stability. Figure A.32 illustrates the best practices in placing BC arcs for culverts.
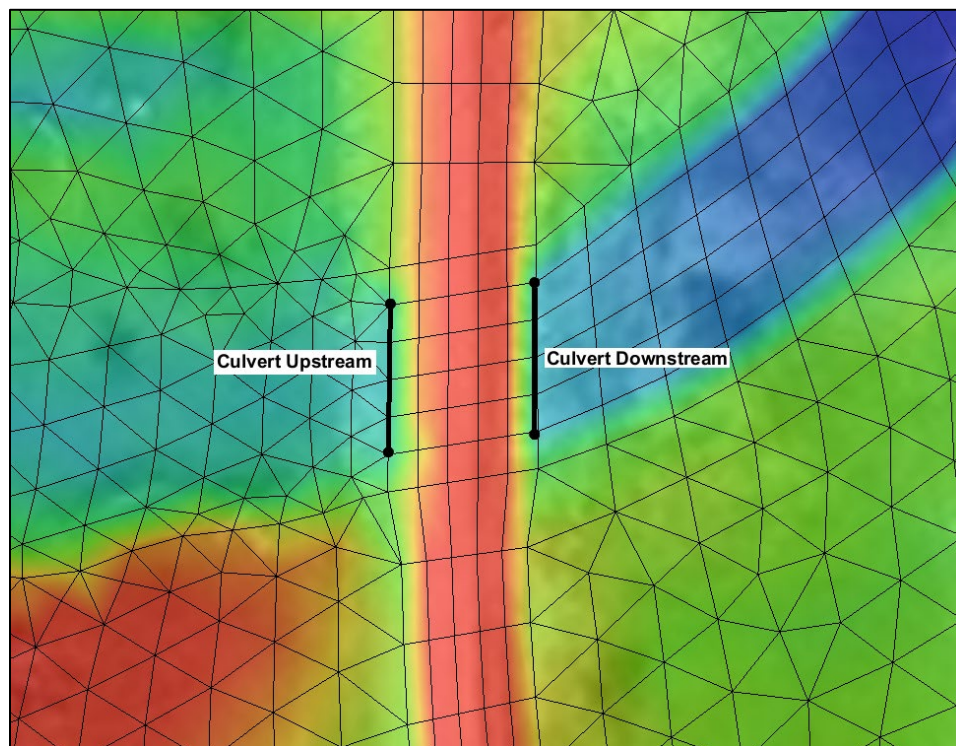

Figure A.32. Modeling 1D culverts in SRH-2D.

Once the upstream and downstream culvert arcs are created, the attributes for the culvert are defined. This is done by selecting both arcs, right-clicking on either arc and then selecting *Assign Linear BC.* Select the Culvert-HY8 BC type, and assign the culvert attributes. In this dialog, it is recommended to select the *Total Head* option for most applications. Most of the attributes are assigned by launching the HY-8 program from within the dialog. The standard HY-8 interface screens appear. Add the culvert attributes in the same way as when using HY-8 independently, except the headwater and tailwater values are not entered since they are extracted from SRH-2D as the model runs. After entering/editing a culvert in HY-8, make sure it is the active (selected) culvert when exiting HY-8.

By default, HY-8 uses an upstream still pool assumption for the computation of the flow in the culvert. No momentum is assumed, although it does have the option to compute flow using the total energy head so that velocity head can be considered. To correctly account for momentum, culverts should be modeled as 2D pressure flow bridges as discussed in the next section, when possible. This is usually only required for larger box culverts in line with the flow.

SRH-2D also supports a generic culvert. In general, this option is similar to the HY-8 culvert option, but the culvert attributes are assigned in the dialog rather than being assigned and computed using HY-8. The generic culvert option has fewer options for types and attributes for culverts, therefore, it is usually recommended to use the HY-8 culvert option.

A.5.4.2. 2D Culverts

Culverts can also be represented in an SRH-2D model by including the base of the culvert in the 2D mesh and mapping the culvert invert elevations (see Figure A.33). The top of the culvert is represented as a pressure flow bridge. The side walls are represented as either very steep side slopes, or by creating a hole in the mesh that specifies vertical walls. Holes in the mesh can create problems if overtopping occurs and the flow is not in line with the culvert, as the holes create infinitely high vertical walls and can impede transverse overtopping flow.
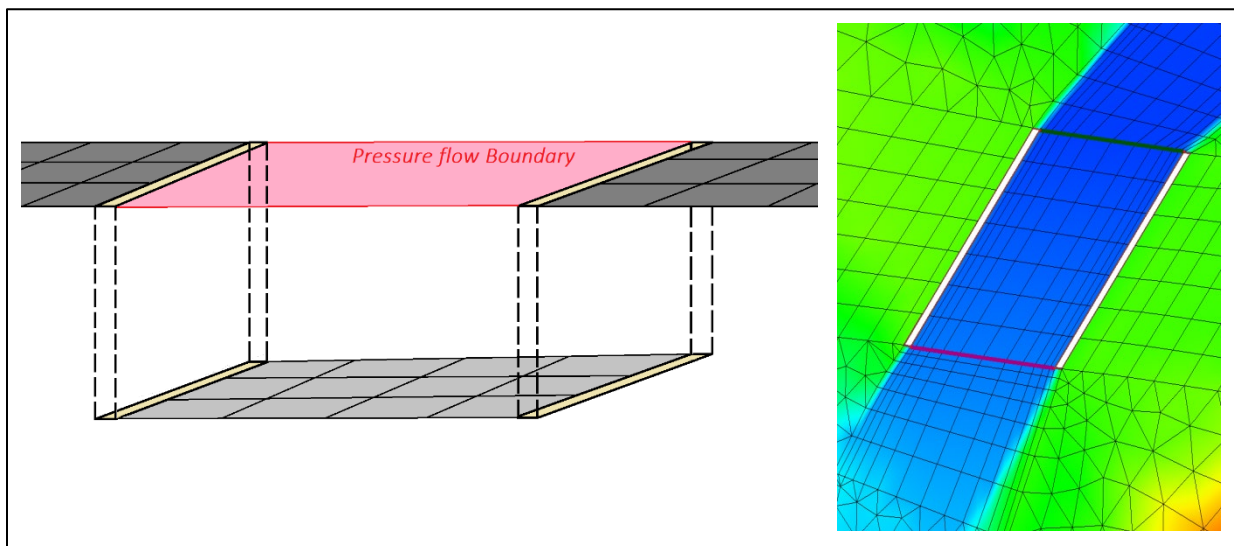

Figure A.33. 2D culverts for SRH-2D.

This method of modeling culverts does not account for head losses that are specifically associated with the inlet edge configuration, but it does provide a better representation of the flow contraction losses at the inlet and accounts for the transfer of momentum. If the inlet edge losses are

significant for a given case, this method may underestimate the head loss. If calibration data is available, additional head loss may be added with a Wall-type boundary condition placed along each of the culvert walls. The roughness of the walls may be increased until calibration is achieved. Additionally, Manning's n value of the channel bed may be adjusted as part of the calibration.

A.5.5. Weirs

Weir flow can exist in many models at roadways or embankments or actual weir structures. Weirs can be represented either in the 2D mesh or as a separate 1D structure. Most often, it is recommended to represent the weir in the 2D mesh. SRH-2D handles transcritical and supercritical flow well, so flow can be properly computed by having the flow go over an embankment and down the side slope through supercritical flow then back to subcritical flow.

The elevations defining the roadway or embankment should be included in the mesh. The mesh elements should be configured to capture the details of the embankment elevations. Usually, this is done by creating arcs representing the toes of the embankment then creating a "patch" mesh of at least 4, elements between (see Figure A.34).
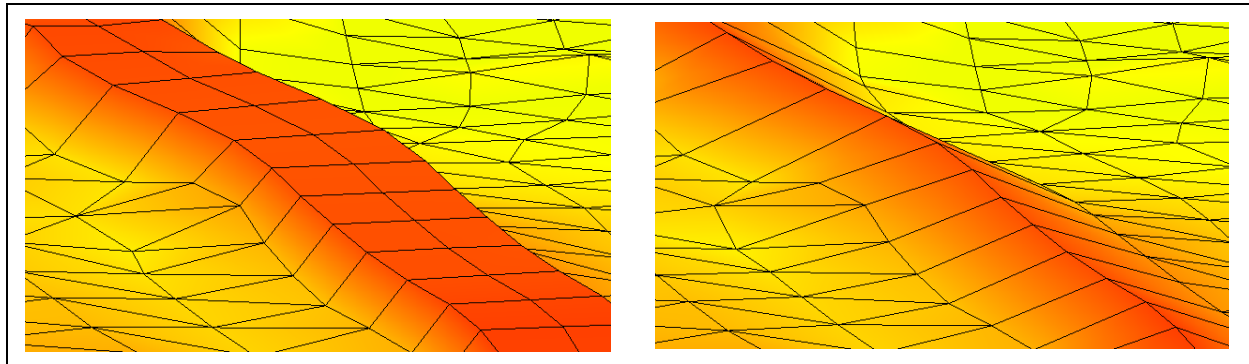

Figure A.34. Meshing to capture embankment elevations.

This results in rectangular elements parallel to the embankment toes and creates enough nodes up and down the embankment to capture the elevation details. For embankments with a flat top or roadway, it is recommended that a full element represents the roadway top elevation to capture the roadway surface elevation. However, SRH-2D properly prevents flow across the element even if only the element edge captures the peak elevation of the embankment as shown on the left in Figure A.34. This is a reasonable approach if it is not necessary to capture details of the flow characteristics of the embankment.

The 2D weir approach discussed in the previous paragraph for roadway embankments can be used for other types of weir structures, as they often are a simple dam and spillway structure. The exception would be a sharp-crested weir that has flow separation as the water goes over the weir. For this type of weir, a 1D weir structure can be used to represent the weir.

As with other 1D structures, the 1D weir structure is defined in the SRH-2D BC coverage using two arcs placed at the approximate location where the weir flow exits the 2D mesh and the location where the weir flow returns to the mesh. The arc length would represent the approximate length of the weir. Be sure to draw the arcs in the same direction. Once the arcs are created, the weir parameters can be set including the crest elevation, lateral length of the weir, and the weir type. Various weir types may be defined as paved and gravel embankments, single and double-crested,

A.45

sharp and broad crested, as well as user-defined weirs. As with other 1D structures, the area between the BC arcs should be constructed using quadrilateral elements, if possible. Figure A.35 illustrates how a sharp-crested weir might be represented as a 1D structure in SRH-2D.
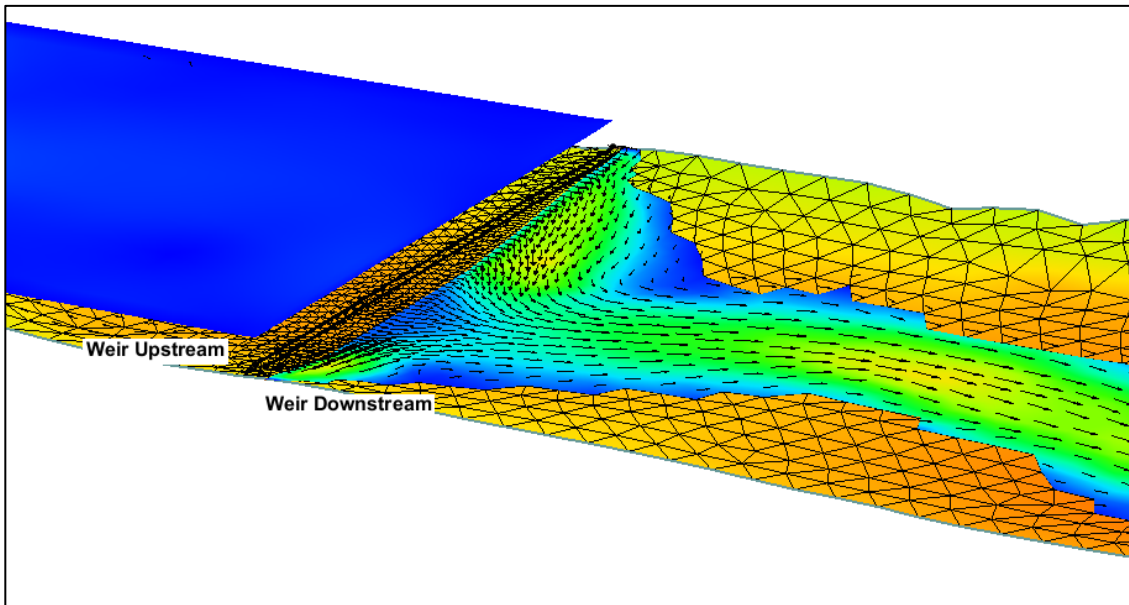

Figure A.35. Photo and oblique view for a sharp-crested weir in SRH-22 (note not the same weir).

A.5.6. Gates

Gate structures are movable barriers that can be used to control the flow of water through or over a dam or embankment. One-dimensional empirical equations have been developed to compute flow rates based on orifice flow through gates. These types of gates are represented in SRH-2D as a 1D structure by adding two BC arcs and assigning attributes for the gate structure in a similar way as described above for culverts. Flow through a gate structure in SRH-2D is calculated as either weir or orifice flow depending on the relative water surface elevations around the gate structure and the pre-defined height of the gate opening.

A.5.7. Obstructions

Another way to represent structures in SRH-2D is to use flow obstructions. Flow obstructions can be used to represent either piers or bridge decks (see Figure A.36). They can also be used to represent any blockage to flow, such as debris that protrudes into the water. Obstructions represent the structure only as a reduction of conveyance area and an additional drag force for the 2D element or elements containing the obstruction. This method is generally recommended only for structures that are not the primary focus of the modeling study. Although obstructions can be used to reasonably represent the energy losses due to structures, they do not represent detailed hydraulics and flow characteristics.
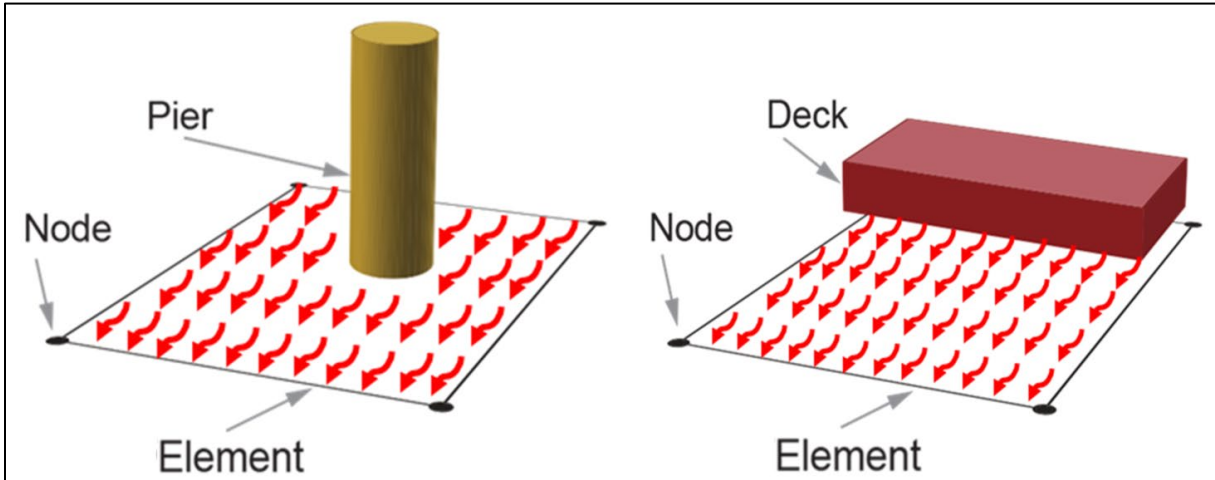

Figure A.36. Pier and bridge deck flow obstructions.

Obstructions are specified in a separate SRH-2D Obstructions coverage. Width, thickness, and a drag coefficient are typically assigned for each obstruction. A porosity, representing the percentage of the projected area of a void, is specified to represent a partial obstruction (porosity of 0.0 for a solid object). Table A.1 shows estimated drag coefficients for a variety of shapes.

Table A.1. Drag coefficients table.

Type of Object	Drag Coefficient	Frontal Area
Long stream-lined body	0.1	
Hollow semi-sphere facing stream	0.38	
Solid Hemisphere	0.42	π / 4 d2
Sphere	0.5	
Cube	0.8	
Thin Disk	1.1	π / 4 d2
Squared flat plate at 90 deg	1.17	
Circular Cylinder	1.0 - 1.3	
Hollow semi-cylinder opposite stream	1.2	
Long flat plate at 90 deg	1.98	

Create a new coverage by right-clicking on Map Data folder in the *Project Explorer* and selecting *New Coverage*. Set the coverage type to *SRH-2D | Obstructions* in the *New Coverage* window. A new name can be specified if desired.

A.5.7.1. Piers

To define a pier obstruction, create a feature point in the Obstructions coverage at the location of the pier. An elevation for the feature point should be entered to define the bottom elevation for the obstruction. For piers, this elevation would be set at a value below the bottom of the channel. To assign attributes for the pier obstruction, select and right-click on the feature point then choose *Assign Obstruction.* This brings up the *Obstructions* dialog. In this dialog, width, thickness, drag coefficient, and porosity are specified. The width represents the width or diameter of the pier. The thickness is the distance from the bottom elevation specified for the feature point to the top of the low chord of the bridge deck. For complex piers with varying widths, the pile cap can be represented by editing the elevations. If the pier width varies vertically it would typically be represented with an average pier width for the portion of the pier that is in the water for the flow condition being studied.

A.5.7.2. Bridge Decks

To define a bridge deck obstruction, create a feature arc at the location of a bridge deck. An elevation for the feature arc is entered to define the bottom elevation for the bridge deck, which is usually the low chord elevation. Flow is unimpeded until the water level reaches this value. To assign attributes for the deck obstruction, select and right-click on the arc, then choose *Assign Obstruction.* In the Obstruction attribute dialog, width, thickness, drag coefficient, and porosity are specified. The width represents the width of the bridge deck measured in the direction perpendicular to the road. The thickness is the distance from the low chord to the top of the bridge deck or railing. Porosity could be used to represent openings in railings, but generally it would not be recommended to go to this level of detail.
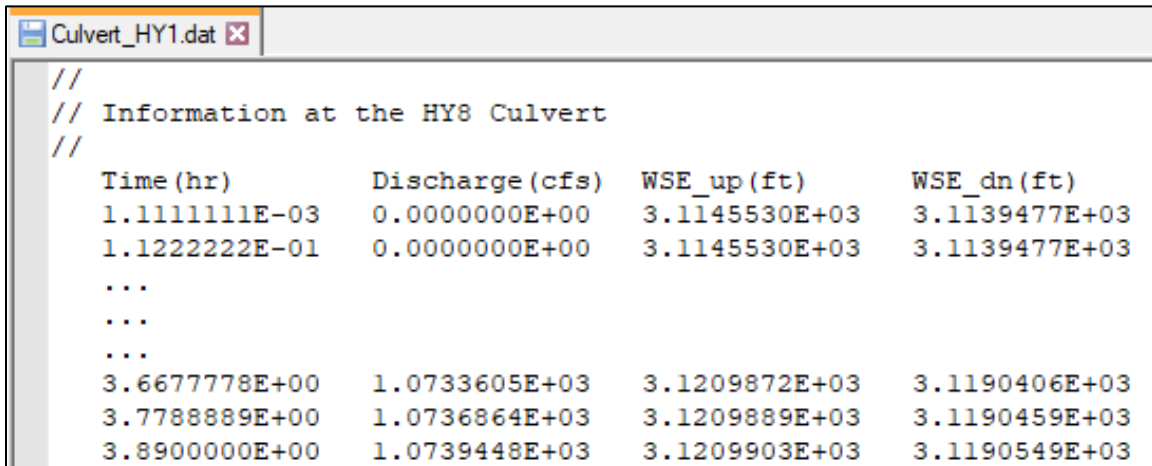
A.5.8. Diversions

Diversions can also be modeled using either a "Link" structure or by specifying an "Internal Sink" boundary condition. Links are used to define flow that can leave the 2D mesh at one location and re-enter at another location. For Links, two arcs are created as is done with other 1D structures, except Links do not disable the 2D elements between the BC arcs. The arcs should be placed in the elements representing the portion of the 2D model where the flow leaves and re-enter the 2D model domain. Select and right-click on one of the two arcs then select the *Assign Linear BC* option. The link attributes for flow characteristics including a routing lag time are then entered.

Flow entering link structures are directly specified as a constant or time varying value or computed using a rating curve or weir definition. The flow through the structure has a specified lag time or a lag time can be computed assuming a full conduit flow approach. If the option to compute the lag time is chosen, the pipe geometry, as well as a roughness value, need to be specified.

An "Internal Sink" boundary condition is used to specify a diversion where flow leaves the 2D mesh but does not return. This is done by specifying a single BC arc, then assigning attributes to define the flow exiting the system at the arc. Alternately a negative number is entered to specify flow entering the model. This same concept can also be applied on the edge of a mesh using an "Exit-Q" boundary condition type.

A.5.9. Output for 1D Structures

A file that includes output data is written for each 1D structure. The report files are assigned a name based on the structure type. The filename has the format *Case_STRUCTn.dat*, where *Case* is the case name specified in the output control, *STRUCT* is a character string identifying the type of structure and *n* is a sequential integer beginning with 1 for each structure in the model in the order they were created. Figure A.37 shows the output for an HY8 Culvert.

```
Culvert_HY1.dat ☒
//
// Information at the HY8 Culvert
//
    Time(hr)          Discharge(cfs)   WSE_up(ft)       WSE_dn(ft)
    1.1111111E-03     0.0000000E+00    3.1145530E+03    3.1139477E+03
    1.1222222E-01     0.0000000E+00    3.1145530E+03    3.1139477E+03

    ...

    ...

    ...
    3.6677778E+00     1.0733605E+03    3.1209872E+03    3.1190406E+03
    3.7788889E+00     1.0736864E+03    3.1209889E+03    3.1190459E+03
    3.8900000E+00     1.0739448E+03    3.1209903E+03    3.1190549E+03
```

Figure A.37. Sample structure output file.

The values in this file are output once every 100 time steps. The time and the discharge (total flow through the 1D structure) is reported for all structure types, but each has different additional information reported relating to the structure. The bullets below show the file names and output data (in addition to total flow) for each type of 1D structure:

- HY8 Culvert – casename_HYn.dat
    - WSE upstream, WSE downstream
- FST Culvert – casename_CULVn.dat
    - WSE upstream, WSE downstream, invert elevation, overtopped (True/False), inlet controlled (True/False)
- Weir – casename_WEIRn.dat
    - WSE upstream, WSE downstream, crest elevation
- Gate – casename_GATEn.dat
    - WSE upstream, WSE downstream, crest elevation
- Overtopped Pressure Flow – casename_INTERNALn.dat
    - WSE, crest elevation

A.6.   MODEL CALIBRATION & SENSITIVITY TOOLS IN SMS

Model calibration is the process of modifying model inputs so that model-computed values match the observed values from an observed event within a specified tolerance. Figure A.38 shows a sample mesh with observed water level data points and computed water surface isolines (linear contours). The contours represent the computed water surface solution from the numerical model. The observed data points are locations of observed (field recorded) water level values and represent the actual solution.
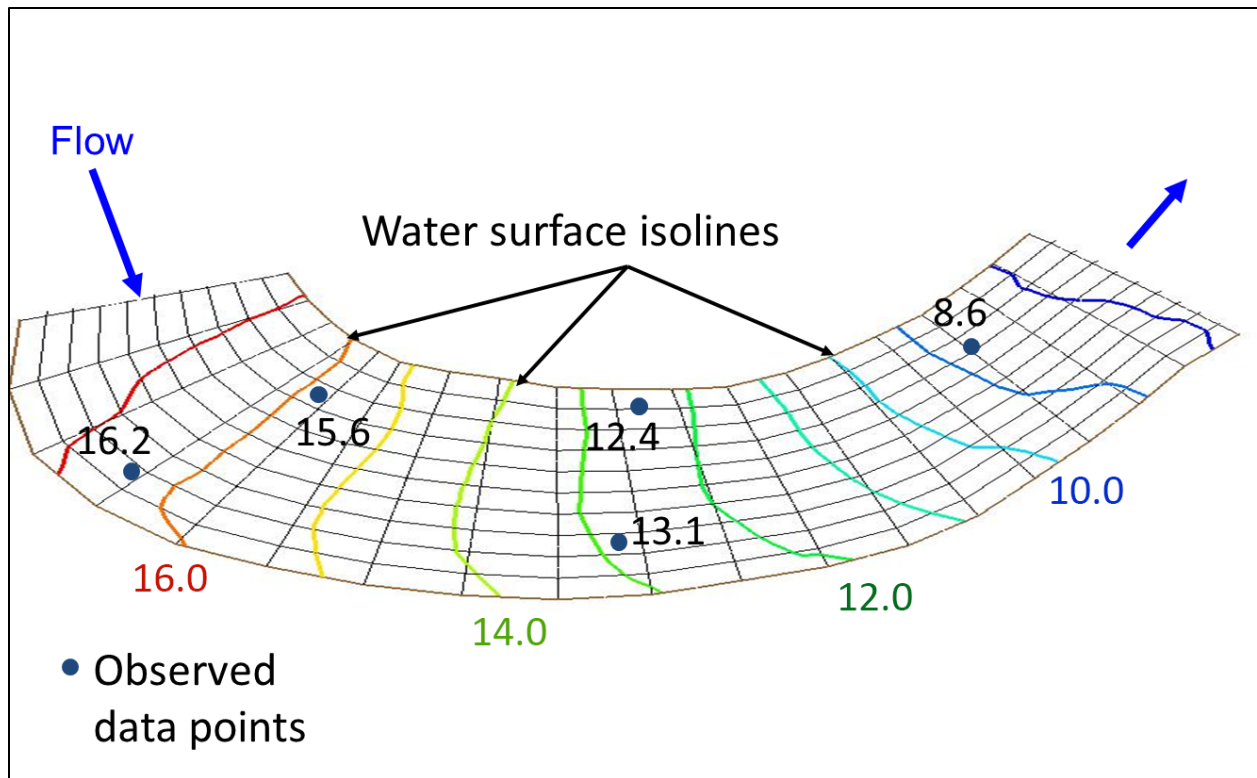
Figure A.38. Observation points on a mesh.

To calibrate a model, a calibration simulation is created and run representing the flow conditions for the observed event. Data measured in the field for this event is read into SMS and compared to the computed values. Roughness or other parameters are then varied, and the model is run again to better match the observed data. This process is repeated until the computed values match the observed values within an acceptable tolerance.

SMS has tools to assist the user with the calibration process. These include tools to import observed data, extract computed values, and visualize the calibration data with calibration targets and plots.

A.6.1. Observation Coverage

Observation data used for model calibration is managed using a separate coverage called an observation coverage. This coverage has the capability to store observed data values at observation points or arcs for comparison against computed values at the same locations. An observation coverage is created by right-clicking on the Map Data folder in the *Project Explorer*, selecting "New Coverage," and selecting the *Observation* coverage type. A new name can be specified if desired.

A.6.2. Defining Observation Points

To define observation points, create a feature point in the Observation coverage at each location where observation data is present. Once the points are created, either double click on one of the feature points, or select *Feature Objects | Attributes* to launch the *Observation Coverage* dialog as shown in Figure A.39.
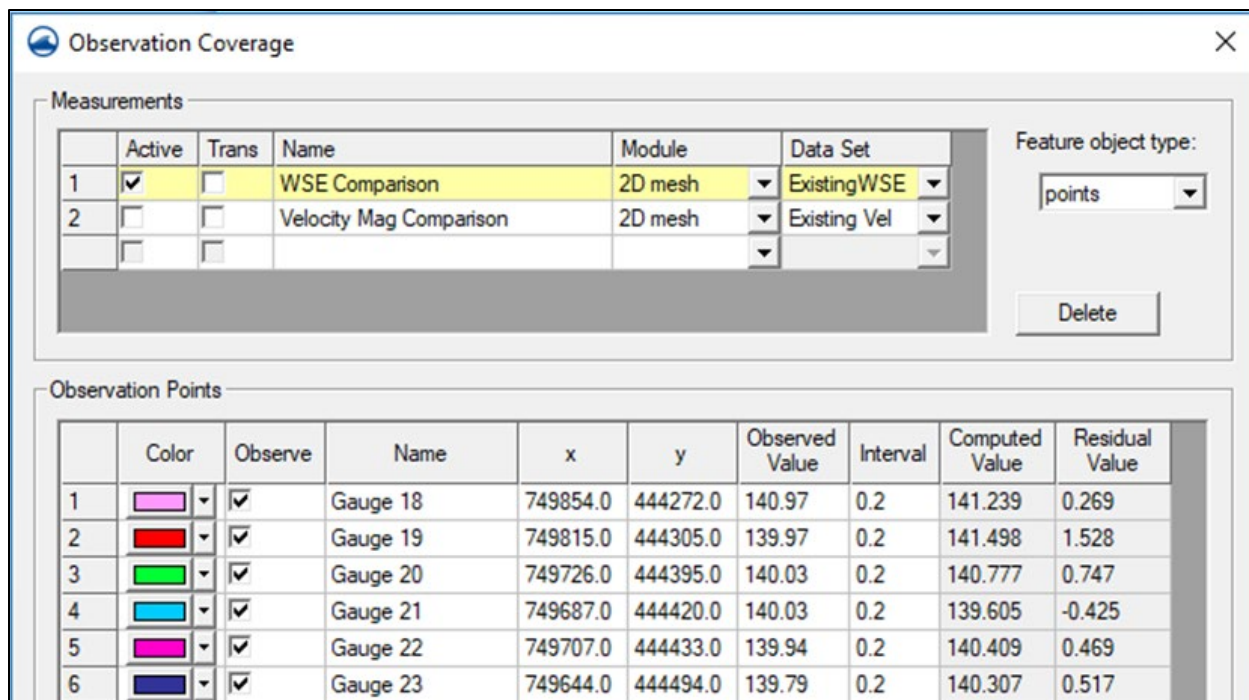
Figure A.39. Observation coverage dialog.

The Observation Coverage dialog contains two main sections: Measurements and Observation Points. The Measurements section is used to select the computed dataset that is compared with the observed data. Multiple measurements can be specified, but only one measurement at a time is active. Transient (time series) measurements can also be specified.

In the *Observation Points* section, the locations of observation points are displayed. The observed value for the measurement is entered for each observation point. An interval is also entered to specify the acceptable error tolerance for calibration. These values can be entered directly, cut and pasted from a spreadsheet, or imported using the File Import Wizard in SMS.

SMS extracts computed values by interpolating the model results in the specified dataset for each measurement at the location of each observation point. SMS also computes the residual value as the difference between the *Computed* and *Observed* values.

A.6.3. Defining Observation Arcs

Observation arcs are defined in SMS to represent measured flow data. This is done in the same *Observation Coverage* dialog except "arcs" is selected as the *Feature object type* in the upper right of the dialog rather than "points." Observation arcs are usually used to compare a flux (flow) across a portion of the model domain. Typically, the arcs would be digitized in the Observation coverage, then one of the arcs would be double-clicked to enter the dialog.

As with observation points, a measurement is defined in the *Measurements* section. Usually, this is a flux measurement. This is done by selecting "2D mesh" as the module and selecting "Water_Depth" as the *Scalar Data Set* and "Velocity" as the *Vector Data Set.* SMS computes total flow across the arc using the element widths, average element depths, and average element velocities summed up for all elements crossed by the arc.

Observation values are entered in the *Observation arcs* section along with a confidence interval as is done for observation points. The *Computed Values* and *Residual Values* calculated by SMS are also displayed in the section, as shown in Figure A.40.
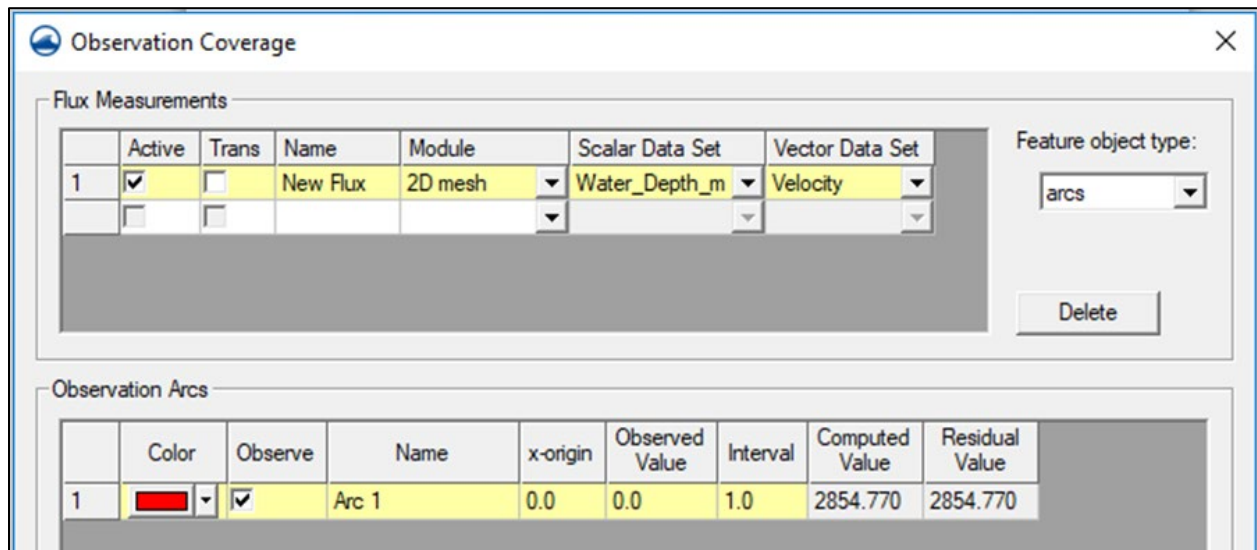

Figure A.40. Computed and residual values at observation points

Calibration targets are a graphical representation of how the computed solution compares with observed data at a specific location. For each observation point, SMS displays a graphical target indicating how well data is matched at each point. Figure A.41 illustrate how the calibration plots are displayed in SMS. Display of calibration targets in observation coverages is turned on in the Map section of the Display Options dialog.
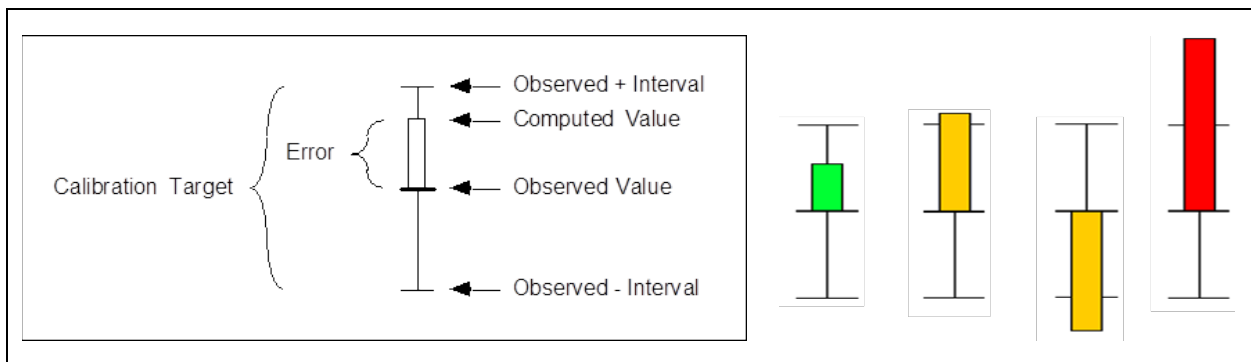
Figure A.41. Calibration targets.

The target shows whether the computed value is higher or lower than the observed value and by how much. If the colored bar is above the center of the target, this indicates that the computed value is higher than the observed value and vice-versa. Green targets indicate error within the interval specified. Yellow targets indicate an error greater than the interval. Red targets indicate error greater than two times the interval.

A.6.4. Calibration Plots

Several different calibration plots can be generated in SMS. These plots can be useful to visualize the data if multiple observation data points are available. The plots are created by selecting the *Display | Plot Wizard* command. If data is available in an observation coverage, a Computed vs Observed Data plot can be created displaying a point with an X value corresponding to the observed value and a Y value corresponding to the computed value for each observation as shown in Figure A.42.
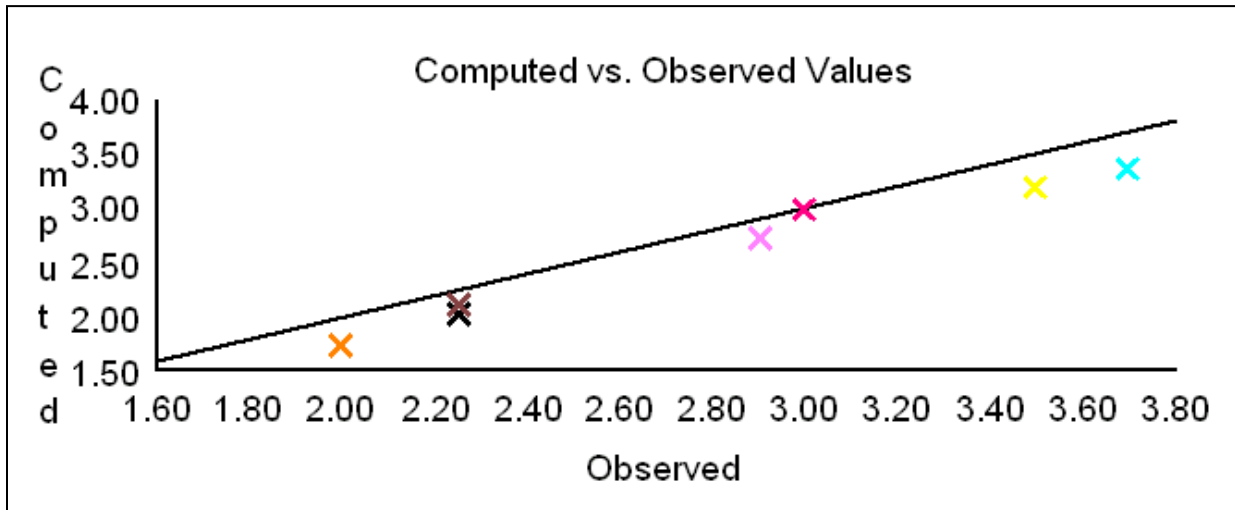
Figure A.42. Computed versus observed calibration plot.

A 1:1 match line is plotted to help visualize how far the computed data varies from the observed. Other plot options include a plot of Residuals vs. Observed Data, Error vs Simulation, Error vs Time Step for transient calibration, and a table of the Error Summary statistics. The *Error versus Simulation* plot is used to track the improvement in the calibration as the parameters are varied in multiple simulations as shown in Figure A.43.

Figure A.43. Error vs. simulation plot.

A.7.   POST-PROCESSING AND GRAPHICAL VISUALIZATION TOOLS IN SMS
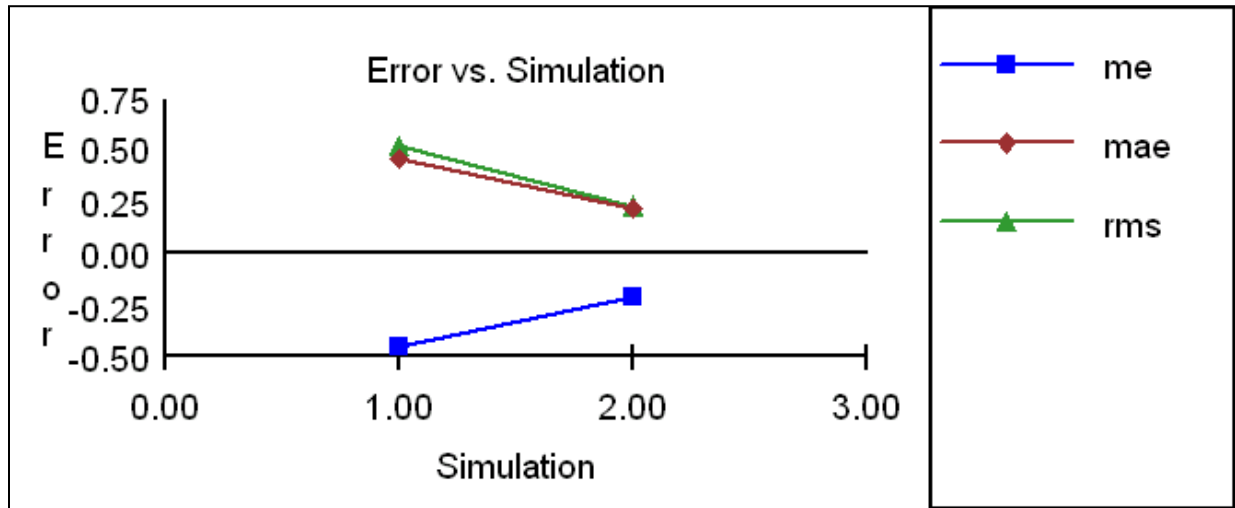
Once a model is run in SMS, the results are reviewed using the post-processing visualization tools in SMS. SMS has a set of visualization tools that allows modelers to review their model results effectively. This section describes the various post-processing capabilities and provides direction on how to use the tools.

A.7.1. Datasets

Solution files contain data values at each node in the mesh. A set of values is called a dataset. Datasets consist of a single scalar value or vector quantity at each node in the mesh.

Datasets that are read from an SRH-2D solution file and include values for water level, depth, velocity magnitude, Froude number, and bed shear stress. Solution datasets are organized into folders and displayed in the *Project Explorer* under Mesh Data (see Figure A.44).
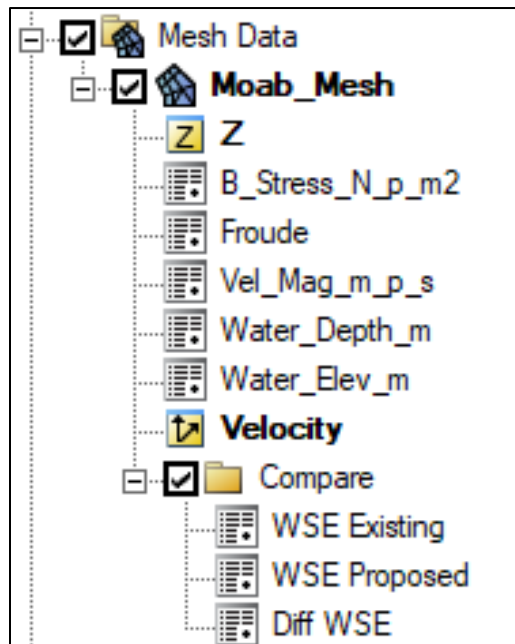
Figure A.44. Mesh data folders.

Datasets may be transient or steady-state. Transient datasets include one value for each node for each output interval specified. The data from the active dataset at the selected output time is displayed in the main display. The active dataset is set by selecting the dataset in the *Project Explorer*. Only one scalar and one vector dataset can be active at a time for a specific module. The name of the dataset that is currently active has a name with bold text.

A.7.2. Dataset Toolbox

The Dataset Toolbox in SMS is a suite of tools used to manipulate and create new datasets based on existing datasets. Figure A.45 shows the *Dataset Toolbox*. Please note this feature is not available in the community version.

Figure A.45. Dataset toolbox.

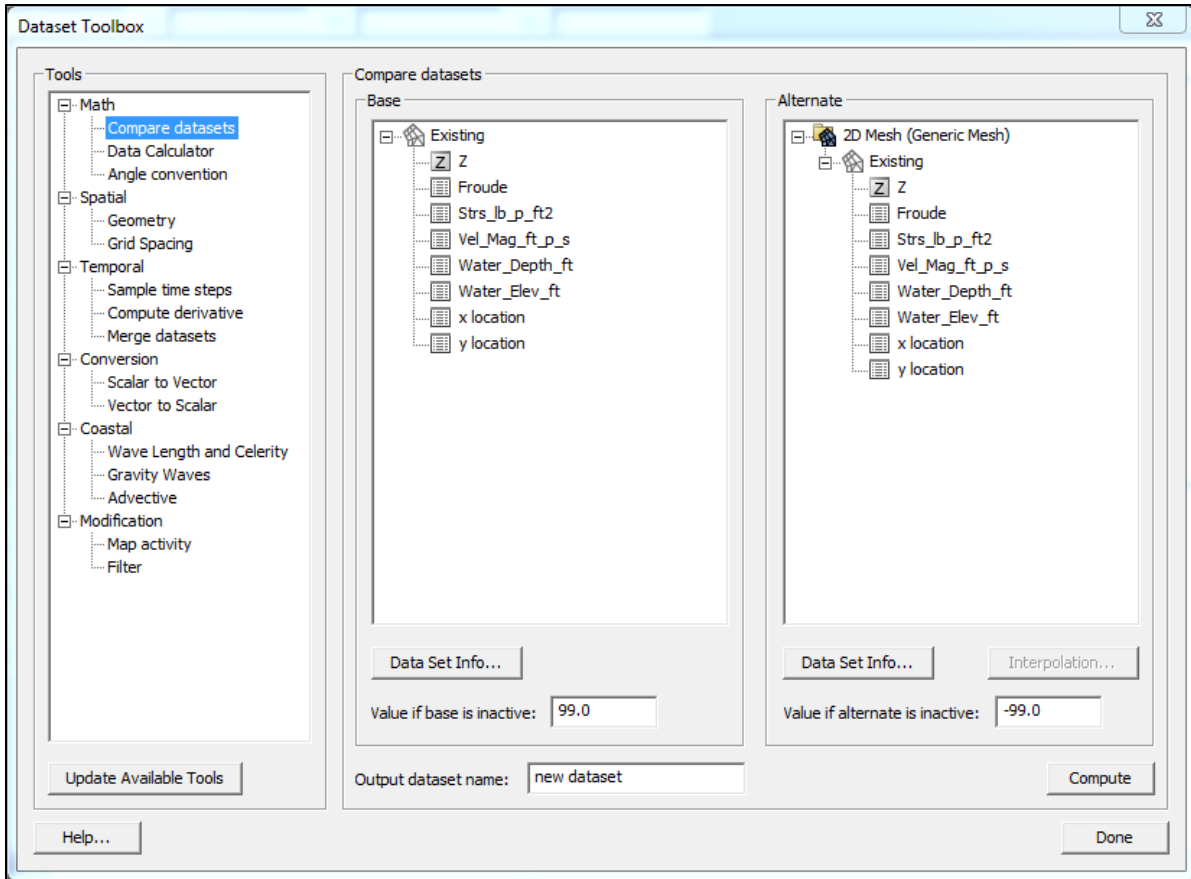The Data Calculator is the most commonly used tool in the *Dataset Toolbox*. It operates similar to a regular calculator except the variables are datasets instead of numerical values. Each dataset has a label–usually numerically beginning with d1 (dataset1). The dataset labels are used as the variables in the equations created in the calculator field. The output from the Dataset Calculator is a new dataset that can then be visualized in the SMS display. Examples of common uses of the data calculator would be to calculate a difference dataset by subtracting two datasets for different simulations or creating a unit flow (q) dataset as the product of the velocity and depth datasets. Other tools in the Dataset Toolbox include tools for filtering, mapping activity, comparing, and resampling datasets. Figure A.46 shows the Data Calculator options in SMS.
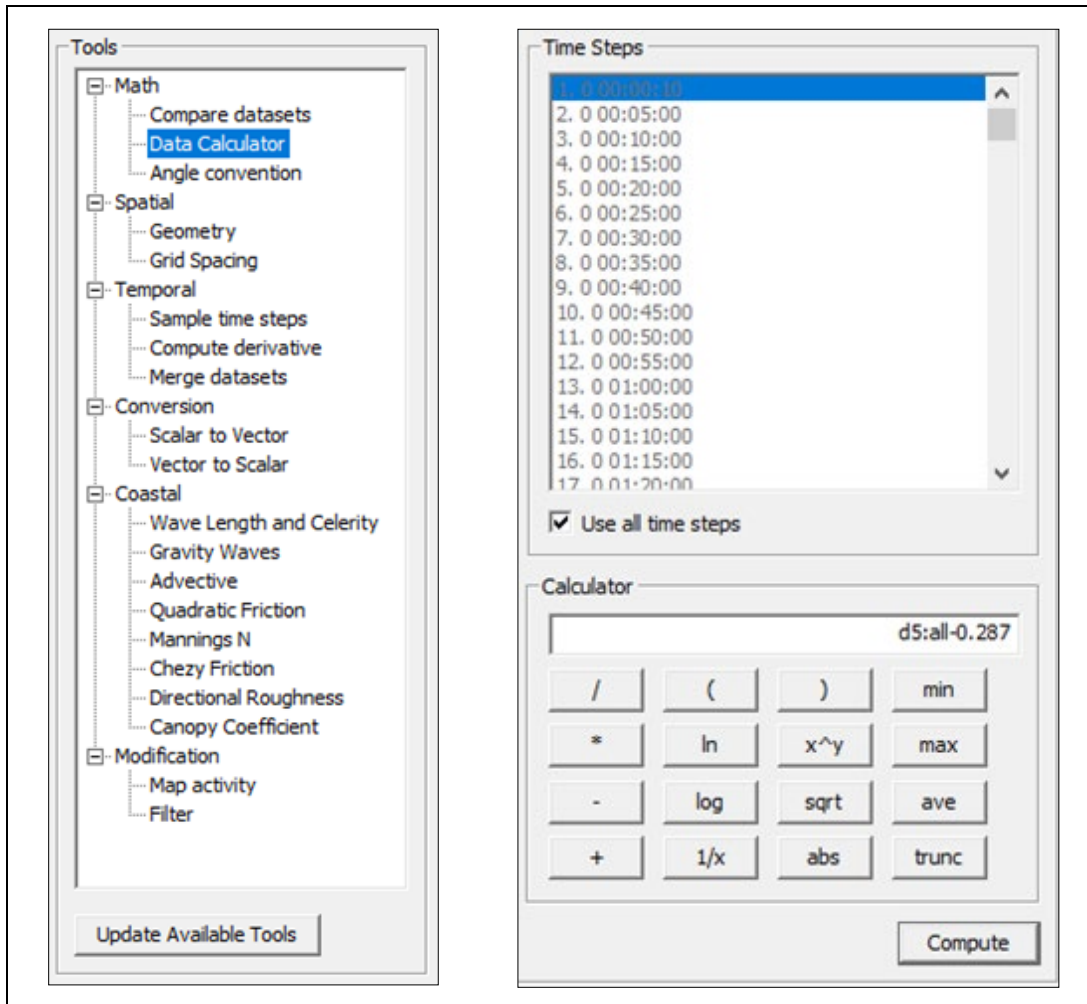
Figure A.46. Data calculator.

A.7.3. Zonal Classification

The Zonal Classification tool in SMS creates polygons in the Map Module for which the data within the polygon(s) meet user-specified criteria. The Zonal Classification tool is accessed using the *Data | Zonal Classification* command. Figure A.47 shows the *Zonal Classification* dialog.

One or more zones are specified in the *Zones* section, each with its own set of criteria based on the available datasets. In the *Criteria List* section on the right, one or more constraints for each zone are specified to control the creation of the polygon zones. Selecting *OK* creates a new coverage with a polygon or polygons enclosing the area or areas in the datasets meeting the specified criteria.

Examples of applications for the *Zonal Classification* tool include identifying areas where a minimum depth and maximum velocity for fish passage evaluation are met, or areas where maximum water depths and velocities are exceeded to quantify flood risk. The *Zonal Classification* tool is also often used to create a polygon representing the flood inundation extents by setting a criteria-of-depth greater than (or near) zero.

A.56
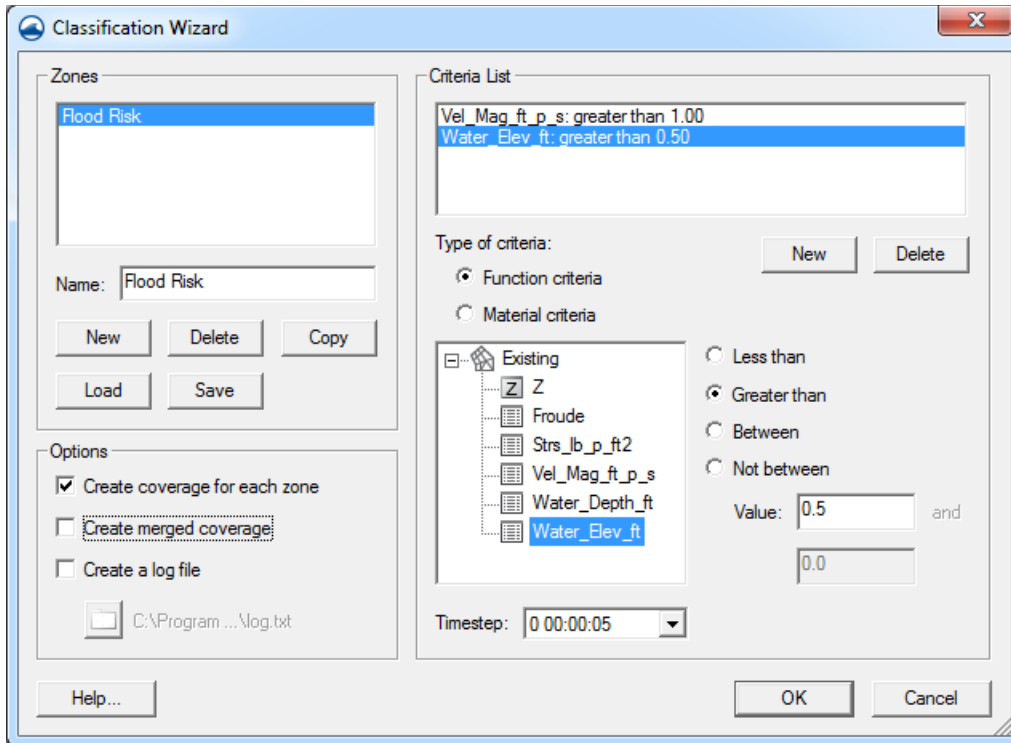
Figure A.47. Zonal classification.

A.7.4. Contours

SMS has tools to control the display of contours. Contour options are accessed using the *Contours* tab in the *Display Options* dialog. There are numerous contour display options in SMS, ranging from basic linear contours to color-filled contours with custom palettes, transparency, specified ranges, and labeling. Figure A.48 shows the dialog for modifying the contour options.

By default, dataset contours are displayed using the global contour options. Global contour options can be overridden with dataset-specific contour options. To set dataset-specific contour options, right-click on a dataset and select the *Dataset Contour Options* command. To revert to using the global contour options for a dataset, select the *Clear Dataset Contour Options* command in the same right-click menu.

Figure A.48. Contour options dialog.

A.7.5. Vectors

Vector arrows are a visual representation of 2D vector data (Vx, Vy). SMS has tools to control the display of vectors. Vector options are accessed in the *Vectors* tab in the *Display Options* dialog. There are numerous vector display options in SMS, ranging from fixed-sized arrows at every node to varying size and color of vectors. A common option selected is to specify that the length of the arrow be scaled to the velocity magnitude under the *Shaft Length* options.

It is often helpful to change the *Vector Display Placement* option from the default (displaying one vector at each node or element) to specifying the "on a grid" option. Using this option sets a vector spacing based on pixel size. As the zoom level changes, the vector spacing remains constant, and vectors are always visible and reasonably spaced. Figure A.49 shows the dialog for modifying the vector options.
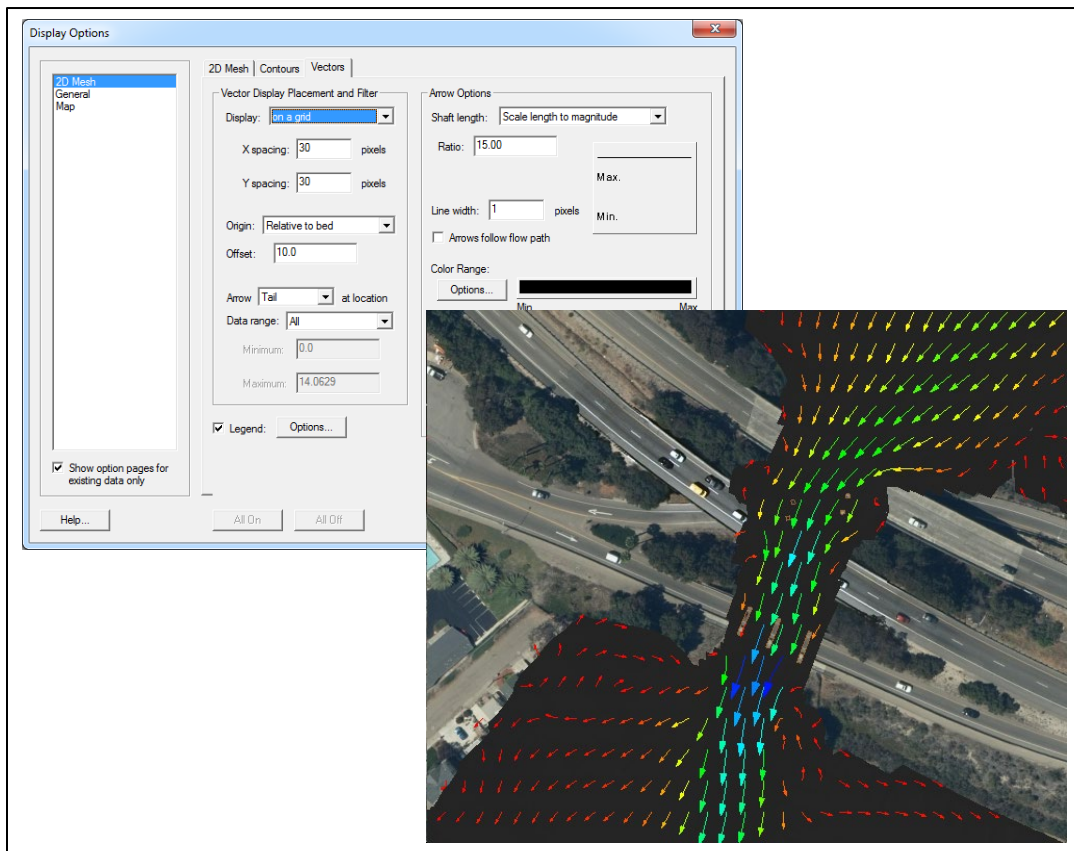
Figure A.49. Vector display options.

A.7.6. Profile Plots

2D plots can be created from various model results. *Observation Profile* plots are the most common. These are plots displaying data along an arc created in an Observation coverage. The method for creating Observation coverages and observation arcs are described previously in Section 4.2. However, these tools can also be used to extract data from the model along arcs.

To create Observation Profile plots, create one or more arcs in an Observation coverage. A plot is generated using the *Display | Plot Wizard* command. The Plot Wizard allows the user to set preferences for the plot, including selecting which arcs are used to create the plot, what datasets and time steps to plot, etc. Figure A.50 shows the *Plot Wizard* dialog options, and Figure A.51 shows sample *Observation Profile* plot.

The *Time Series* plot is another type available in the *Plot Wizard*. This plot type generates plots for data values at a single observation point over multiple time steps. Various calibration plots can also be created using the *Plot Wizard* as described in the Calibration section above.
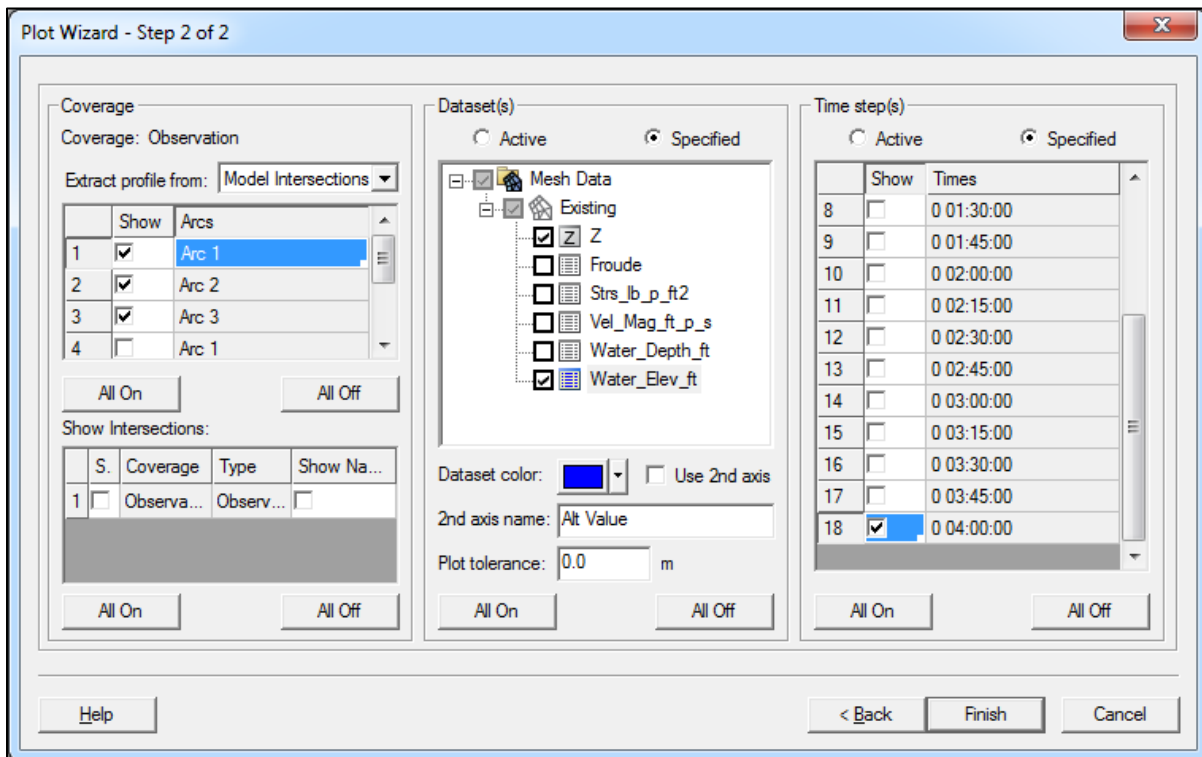
Figure A.50. Plot wizard dialog.
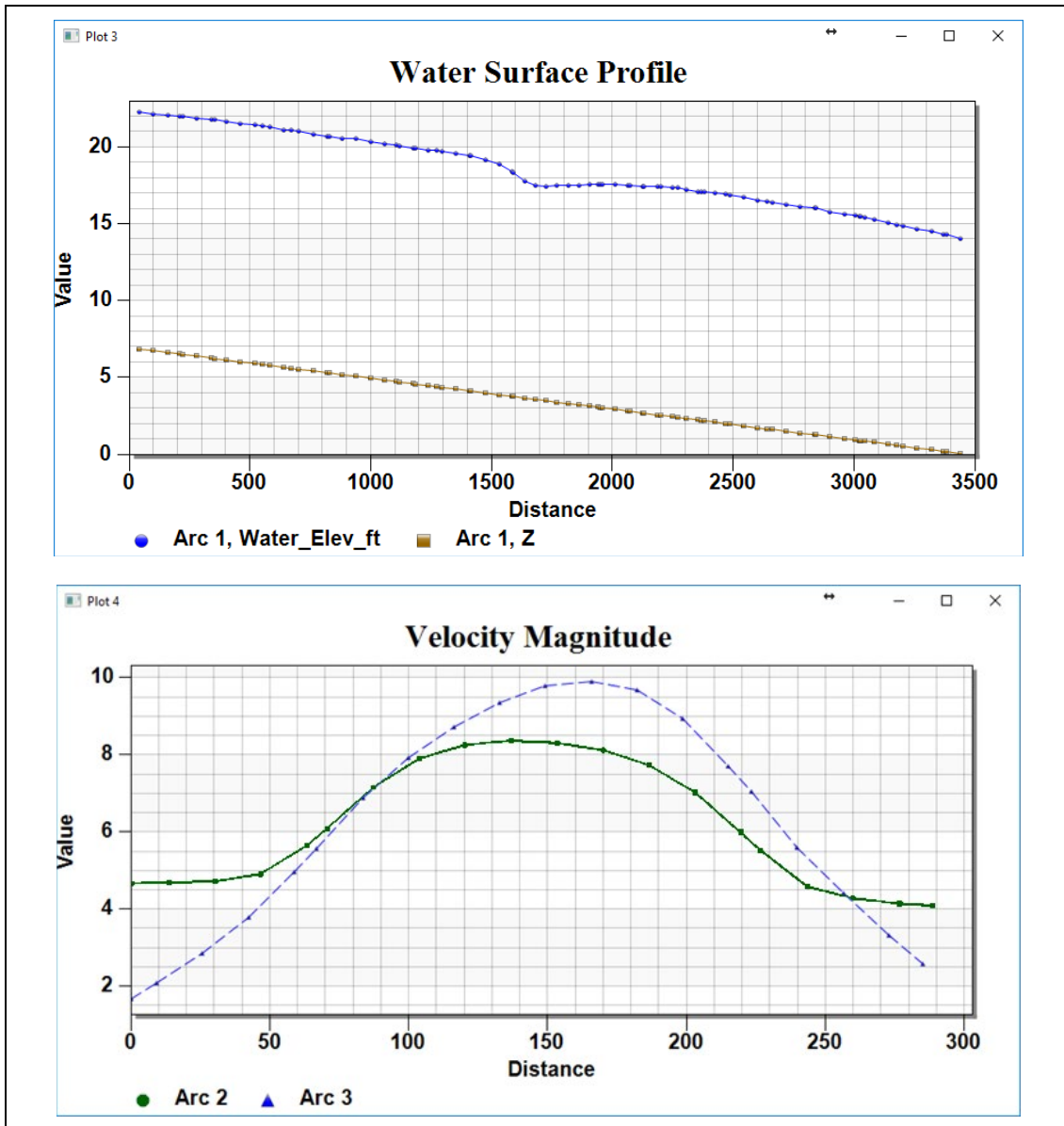
Figure A.51. Observation profile plots.

A.7.7. Summary Tables

Summary tables can be created to extract results along cross sections relative to a river centerline. If a 1D model exists, the cross section geometry can be imported and overlaid on the 2D model results. If not, both a 1D Hyd-Centerline and a 1D Hyd-Cross Sections coverage should be created to represent a centerline, banks and at least one cross section.

To create a summary table, right-click on the *1D Hyd-Cross Sections* coverage and select *Summary Table.* Figure A.52 shows the dialog to set the Summary Table options.
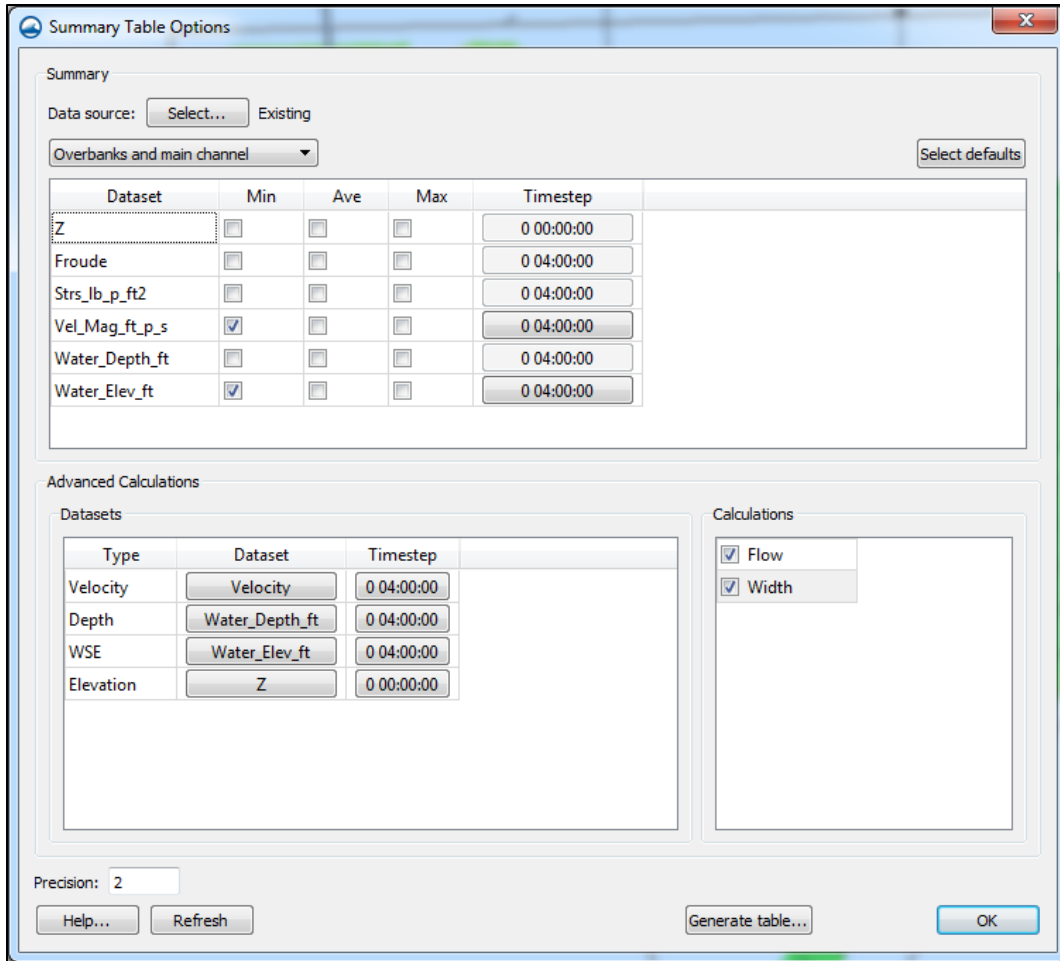
Figure A.52. Summary table options dialog.

Summary Table parameters for each cross section that can be extracted include:

- Cross Section Station based on the distance along the centerline arc
- Flow
- Flow width
- Average, minimum, and maximum values for:
  - Velocity
  - WSE
  - Depth
  - Froude number
  - Bed shear stress
  - Any other mesh or scatter dataset

Users specify whether values are computed for the entire cross section, the main channel only, or the overbanks and main channel separately. Figure A.53 shows a sample set of cross sections in a 1D Hyd-Cross Section coverage alongside a sample Summary Table.
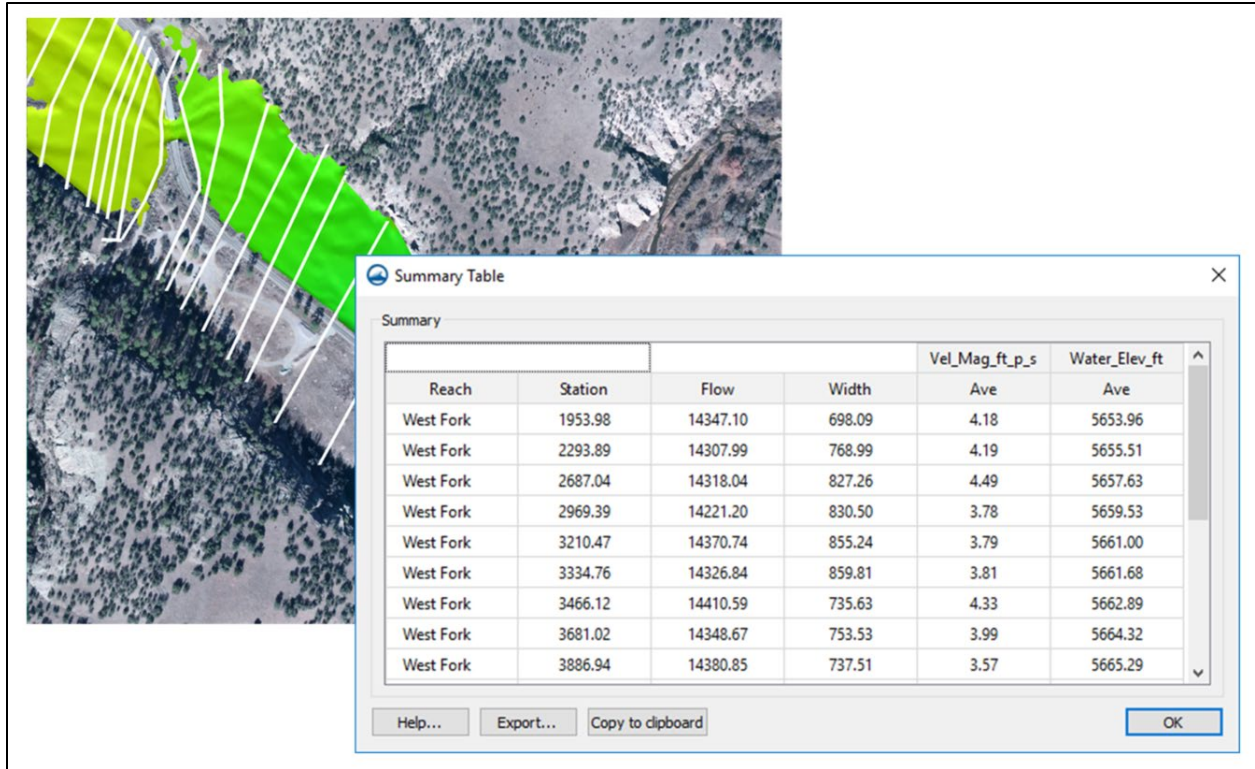
Figure A.53. Summary table display.

A.7.8. 3D Visualization

SMS visualization options include the ability to view model data in 3D since the mesh contains elevations and other datasets that can be used to create 3D surfaces. The model can be viewed in an oblique view to visualize various 3D effects. The Rotate Tool 🔄 on the main toolbar is used to rotate the data objects out of the standard plan view display. The 3D viewing angle and the motion of dynamically rotating the view help to visualize and identify problems with elevation surfaces easily.

A.7.9. Z Magnification

As vertical variations in the terrain can be subtle relative to the X, Y extents of a model domain, it can be helpful to exaggerate the display of elevations. This is done using the Z magnification option, which is set in the *Drawing Options* within the *General* section of the *Display Options* dialog. The default is "Auto z-mag" which sets the z magnification as a percentage of the range of the elevation dataset. Figure A.54 shows this section of the Display Options dialog.
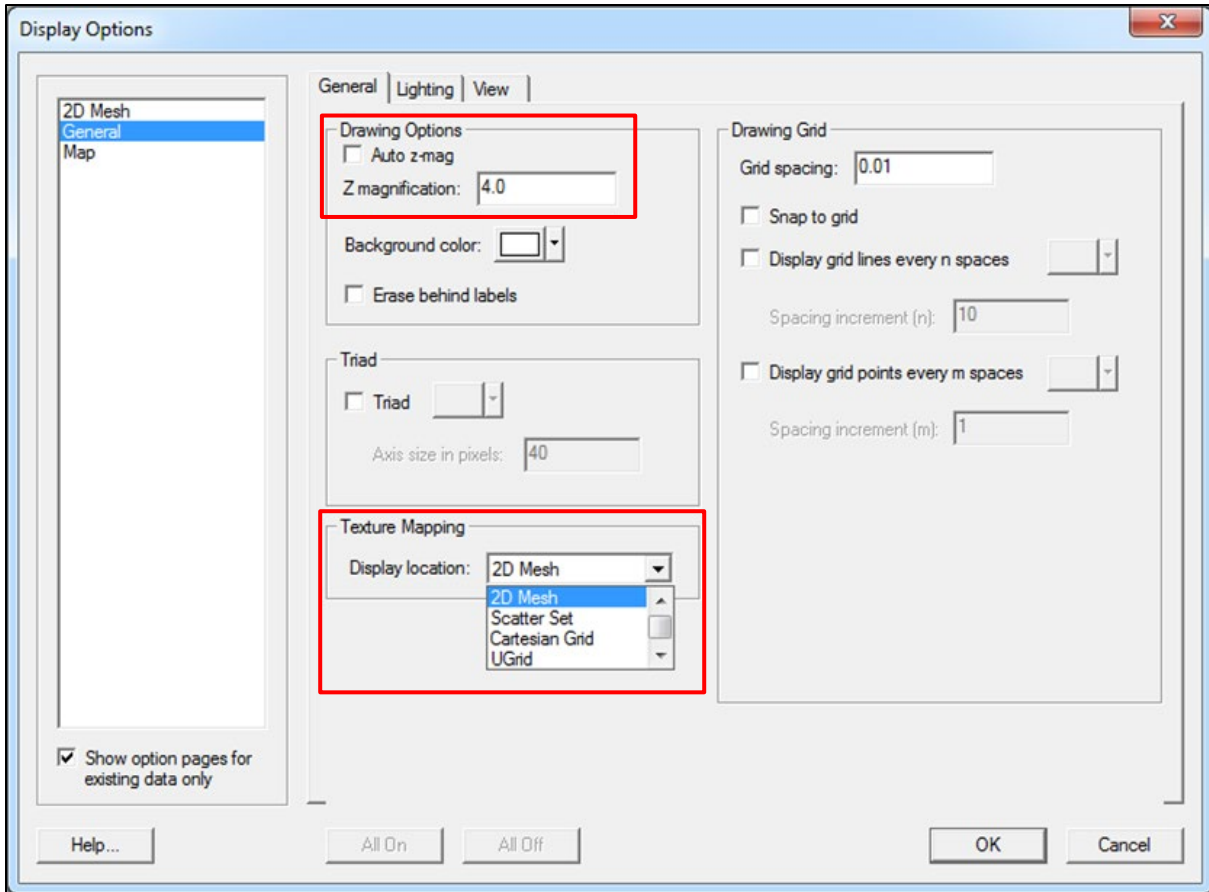
Figure A.54. Display options dialog, showing the General Tab.

A.7.10. Texture Mapping

A texture map is a raster image mapped or "draped" on a surface. Texture mapping is an effective tool for visualizing a study area in 3D. Texture mapping is also accessed in the *General* section of the *Display Options,* as shown in Figure A.54. Images are draped onto surfaces defined by meshes or scatter sets. Lighting and shading are enabled to further enhance the visual appearance of the texture mapped image. Note: Texture mapping is graphics intensive and requires a dedicated graphics card that meets the system requirements found on the SMS webpage. It may not work if the only graphics processing available is integrated graphics (common in laptop computers). Figure A.55 shows an example of a 3D view of a texture mapped image in SMS.

Figure A.55. Texture mapped surface.

A.7.11. Shading

Shading and lighting options are used to enhance the display of 3D data. Shading provides a more realistic representation of elevation differences in oblique views. Shading is accessed from the *Lighting* tab in the *Display Options* dialog. Lighting options allow changes to the light source direction, ambient light, diffuse light, specularity, and shininess. The light source position is the lighting parameter most often adjusted. It is changed by clicking on the dot on the red sphere in the dialog and moving it to a different point on the sphere. This is analogous to the direction from which sunlight would approach the 3D surface of the earth. Figure A.56 shows the *Lighting* options in the *Display Options* dialog, and a sample 3D surface with lighting enabled. The default settings are usually recommended, except that it is often helpful to move the light source location.

Figure A.56. Shading options dialog.

A.7.12. Functional Surface

A Functional Surface in SMS is a representation of a scalar dataset on a surface other than the terrain surface. A Functional Surface is most commonly used to display a dataset at the position of a spatial dataset such as the water surface. Usually, the contoured elevation surface is visible beneath the Functional Surface. Other options include display of a surface offset from the ground surface. A Functional Surface can also be used to create a geometric representation of other scalar datasets such as velocity magnitude, surface pressure, etc.

Functional Surfaces are turned on in the 2D Mesh section of the *Display Options* dialog. In that section, there is an *Options* button next to the Functional Surface toggle. Options for Functional Surfaces include specifying the dataset to define the plotting surface, Z offset options, alternate Z-magnification, color, and transparency. Contours based on any dataset can be displayed on the Functional Surface. Figure A.57 shows the dialog for setting the Functional Surface options and a sample of a Functional Surface with depth contours plotted at the location of the water surface.
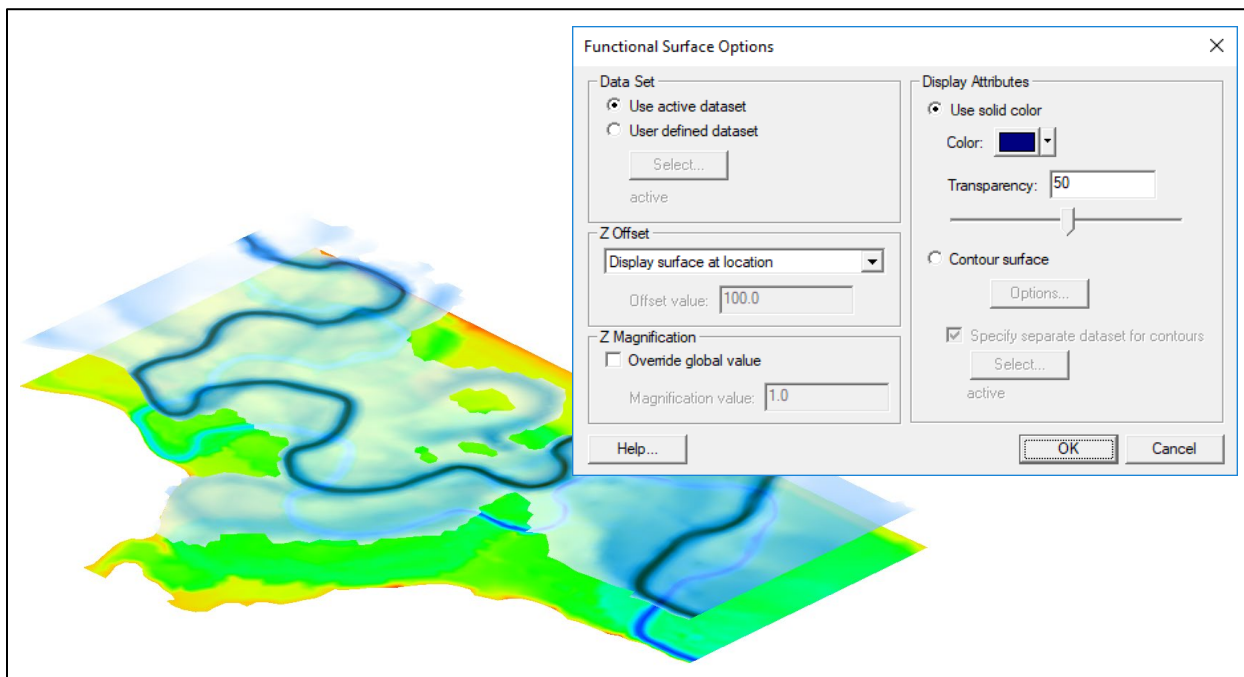
Figure A.57. Shading options dialog.

A.7.13. Animations

Animations or videos of solution data can be created in SMS. SMS has options to create various types of animations that are saved as video files for Windows (.avi files) or Transient Google Earth files (.kmz). The .avi animations can be played using the Pavia video player bundled with SMS or using any standard media player.

The types of animations that can be created in SMS include:

- Transient Data Animations: Show how contours and vectors change with time
- Flow Trace: Visualization of flow field using steady-state or transient velocity vector data
- Drogue Plot: Time series particle trace using vector data

- Plot Window: Series of 2D plots changing with time

Animations are created by selecting the Data | Film Loop command while in the mesh or scatter module. At least one transient dataset has to exist in the mesh or scatter set to create a film loop (animation). The browser button is selected to specify a file name or location different from the default (sms.avi). Options are set for timing and image quality, but usually, the defaults are sufficient. Once SMS has finished creating the animation, it automatically launches using the Pavia application provided with SMS. The saved .avi file can be viewed using any media viewer and embedded in a PowerPoint presentation. This animation can also be saved in a .kmz format for display in Google Earth. Figure A.58 shows the dialog used to set up animations in SMS.
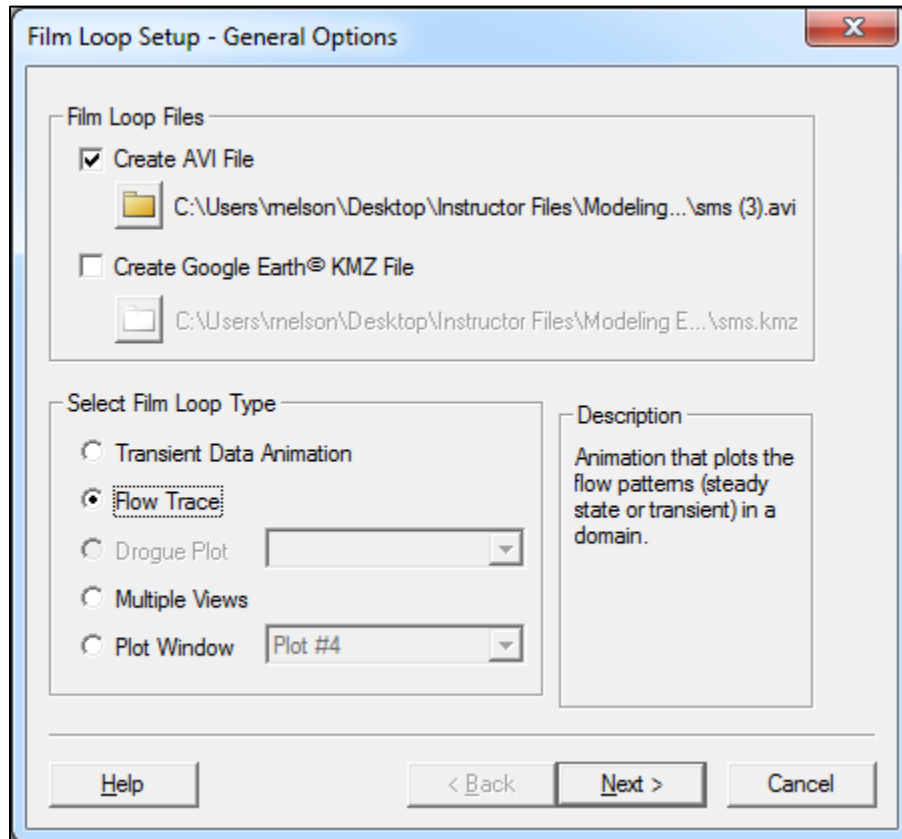
Figure A.58. Film loop setup.

For a Transient Data Animation, solutions containing multiple time steps should be present. The view and display options are set as desired, and then the animation is created by recording a view for each time step. These animations are similar to what is seen on the screen when cycling through the time steps one at a time in SMS.

Flow Trace animations provide a dark background and simulate the dynamic flows by placing massless particles throughout the model domain. Flow trace animations can be created for both steady-state and transient solutions. The animation simulates how the particles would move using the velocity flow field to map the location and speed of the particles. This is analogous to how smoke is used in a wind tunnel to visualize the airflow. This provides a good visualization of how the water moves through the model and the relative velocities of the water at different points in the model. Figure A.59 shows a single frame from a sample flow trace animation.

A.68
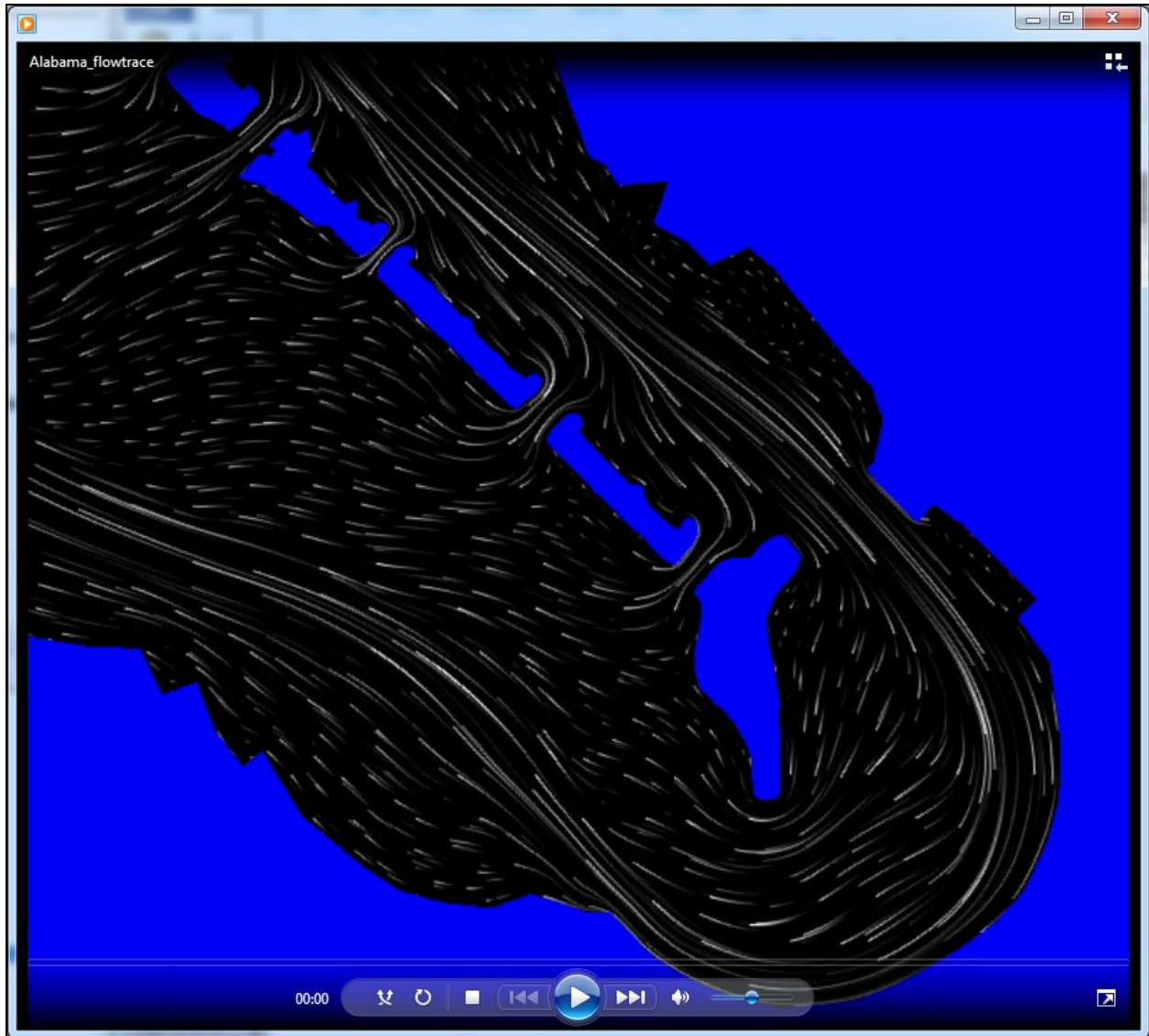
Figure A.59. Flow trace animation.

Drogue Plots are similar to flow trace animations except they allow the user to place specific massless particles called drogues in specific locations. This is often used for particle tracking to determine the location and timing of how contaminants or sediment might migrate. Transient data is required for drogue plots.

Plot Window animations allow for creating animations of plot views for each time step, similar to the transient data animations. This option is only available for *Observation Profile* plots. Any active plot window can be selected.

A.69

A.7.14. Annotations

The use of Annotations in SMS is helpful in labeling and clarifying the data in the display window. It also helps in making the display more presentable for inclusion in report graphics. The Annotations Toolbar contains different tools for creating and editing annotation objects in SMS (see Figure A.60). To access the annotation tools, select the Annotation Module icon from the bottom of the SMS interface.

Figure A.60. Annotation toolbar.

Features include the ability to add text, lines, arrows, ovals, and rectangles to the display. Scale bars, north arrows, and images (logos) can also be added. Once the Annotation Module is activated, the tools for adding annotation objects display in the dynamic toolbar, usually just left of the *Project Explorer*. Annotation objects are created by selecting a tool and then clicking a location or dragging the shape. Usually, there is a prompt to specify whether the object is anchored to "Screen" or "World" coordinates. World coordinates allow the object to zoom and pan with the other objects, whereas screen coordinates fix the object to a certain screen position. For each object, a separate dialog appears to set various options for each type of object.

A.7.15. Exporting Data

SMS allows the user to export data in various formats. The most common method is using a screen capture utility. Other options include:

- *Copy to Clipboard* – Ctrl + C keyboard shortcut. This places the contents of the SMS *Display Window* in the paste buffer.

- Print – Select *File | Print* from the top menu. Prints the contents of the SMS Display Window to a printer.

- Export Image – Select *File | Save As* from the top menu and change the *Save as type* option to .bmp, .jpg, .png image format. Exports an image of the *Display Window* in the specified image format. This option has the advantage of being able to export an image in a higher resolution than the screen resolution.

- ESRI Shapefiles *(.shp)* – Select *File | Save As* from the top menu and change the *Save as type* option to *Shape Files (*.shp)*.

- CAD *(.dxf, .dwg)* – Right-click in the blank space at the bottom of the *Project Explorer* and select *Convert to CAD*. Vector data (linear contours, points, arcs, or polygons) should be present in the SMS project, or nothing is exported.

- Google Earth *(.kmz)* – Select *File | Save As* from the top menu and change the *Save as type* option to the raster or vector Google Earth .kmz format.

Figure A.61 shows an example of an image exported in a Raster Google Earth (.kmz) format.
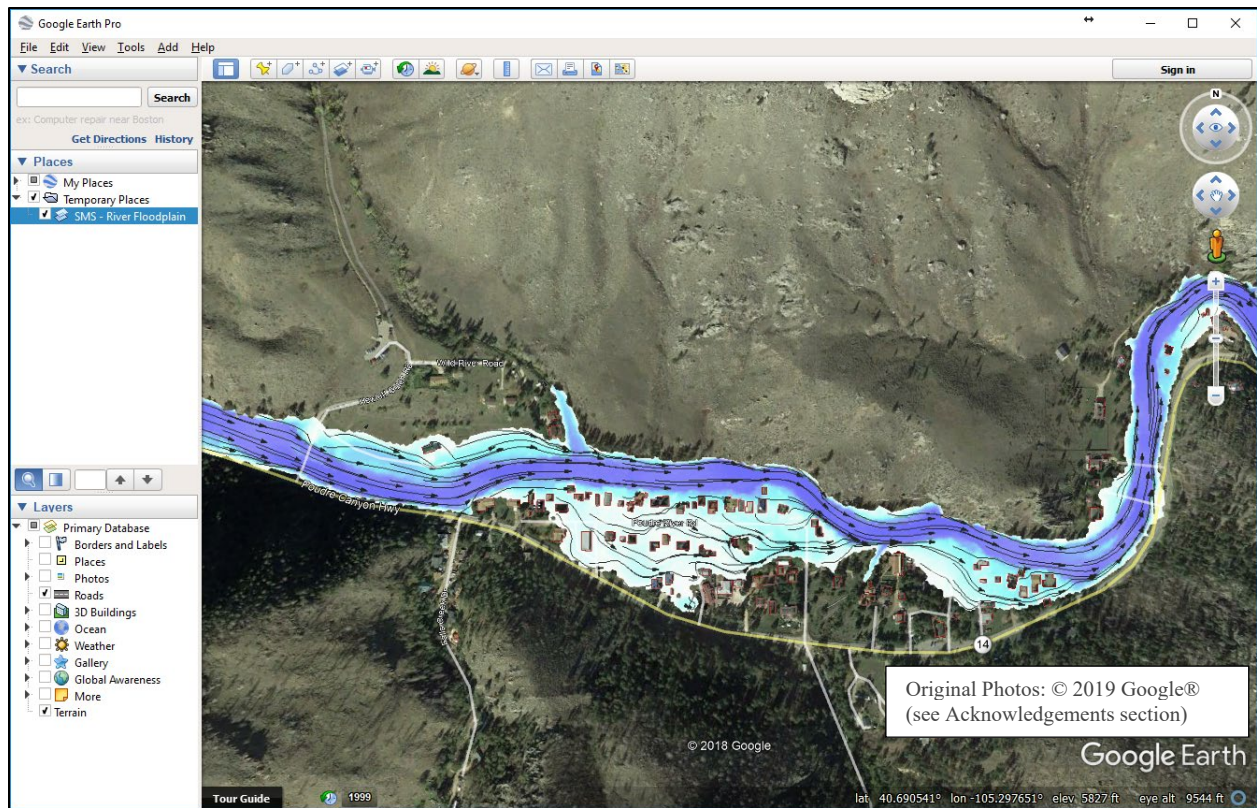
Figure A.61. KMZ file displayed in google earth.

A.8. IMAGE REFERENCES

Image	**Source**
Figure A.3	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
Figure A.5	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
Figure A.7	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
Figure A.18	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
Figure A.35	Photo "Weir Long Exposure" from Pixabay https://pixabay.com/photos/weir-long-exposure-waterfall-water-1903139/
Figure A.48	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
Figure A.49	Background imagery from ESRI World Imagery. https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer

Figure A.52   Background imagery from ESRI World Imagery.
https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer

Figure A.55   Background imagery from ESRI World Imagery.
https://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer

Figure A.61   The original maps are the copyright property of Google® Earth™ and can be
accessed from https://www.google.com/earth (Google, 2019). The Authors
developed the map overlays for this document.

APPENDIX B

SURVEY DATA REQUEST

B.1. SURVEY DATA REQUEST

A quality survey of existing terrain is crucial for producing a correct representation of a project site for 2D model development. Detail for various data sources and modeling requirements are discussed in Chapter 4. This appendix focuses on data collection methods typically performed by a Department of Transportation (DOT) survey crew as well as an example survey request form (Appendix B.2) to effectively communicate the level of survey needed by the modeler to the survey team.

Every site has unique features and challenges that should be evaluated by the modeler prior to completing a survey request. By understanding site features and project goals, the modeler can identify terrain features of the project site that should be collected and then allow the surveyor to determine the most efficient way to collect the requested data. It is advantageous to perform a site visit prior to submitting a survey request. If this is not possible, the modeler should meet with the survey crew on-site at the beginning of the collection period to assist in minimizing the need for additional trips to collect data that may not have been effectively communicated at the beginning of the project. This allows the modeler to identify features, with the survey crew in the field, that should be included in the survey to produce an adequate representation of the project site for a 2D model.

Several considerations are required when creating a survey request. These include, but are not limited to, the extents of the survey, specific features and attributes to collect, and accuracy requirements (see section 4.2.3 in main document). Included in this appendix is an example of a Survey Request Form that could be used to communicate with a survey crew about the extents and features required in the survey. In addition to the form, an exhibit of the project site, preferably with an aerial background, should be attached to the form depicting the various hydraulic features and site information of interest. Modelers should note if existing lidar or other information is available that would be pertinent to the project site.

B.1.1. Project Extents

To determine full project survey extents (described in more detail in Chapter 4), modelers should have a basic understanding of hydraulics at the project site, as well as the purpose of the project. For example, if the project is a floodplain study that uses the Federal Emergency Management Agency's (FEMA) Base Flood Elevations (BFE) to complete a Conditional Letter of Map Revision (CLOMR), or something similar, then the survey extents should encompass the FEMA BFEs and the FEMA mapped floodplain boundary within the project area. If the proposed design includes a channel realignment, the survey extents should include the area where the expected reconstructed channel and its anticipated floodplain would be. To help determine survey extents, it may be useful to conduct a preliminary hydraulic analysis of the project site to understand inundation limits and potential boundary conditions better.

To determine the appropriate upstream and downstream survey extents for a project site, boundary conditions should be taken into consideration (describe in more detail in Chapters 4 and 5). The survey should extend sufficiently upstream and downstream such that the model boundary conditions do not affect model results at the project site. For example, if there is a confluence near the project site, it may be advantageous to collect data at the confluence to correctly model flow through the other waterbody. Alternatively, if the project site includes a large alluvial fan or braided channel, enough survey data should be collected upstream so a proper boundary condition can be defined.

Channel geometry, surveyed breaklines, and points should be collected along the thalweg, bank bottom, bank top, and any additional locations where there is a change in grade. Figure B.1 shows an example of the minimum number of channel features that should be collected.
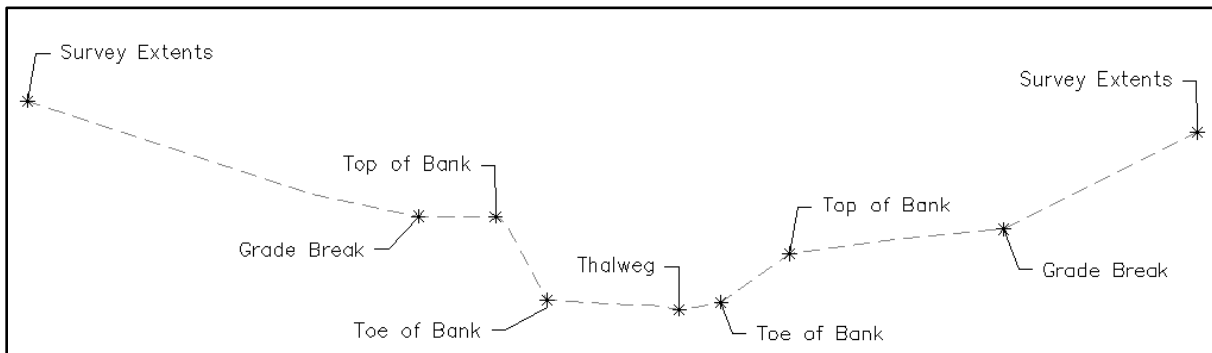
Figure B.1. Example of Point/Breakline Locations That Should Be Collected to Define Channel Geometry.

B.1.2. HYDRAULIC FEATURES

The following sections detail the various hydraulic features that could be present near a project site and key components associated with those features to collect during a site survey. As previously mentioned, a modeler should have enough information related to the project site to be able to communicate with a survey crew what features are required.

B.1.2.1. Bridge

Bridge components that should be represented in a survey, when an existing bridge is present on a project site, are identified below. The level of detail of each component should be determined by the modeler based on the project purpose and site features. For example, if an existing bridge experiences pressure flow with overtopping of the roadway, the full geometry of the bridge should be surveyed to be able to model the flow area under and over the structure. However, if the low chord of an existing bridge is expected to be substantially above the design flow WSE, then the modeler may be satisfied with less detail. Additionally, if the purpose of the project is scour mitigation at an existing bridge with undermined footings, the modeler needs to request a more detailed substructure survey. This may include breaklines depicting the detail of scour holes, the amount of exposure along a pier, or other substructure details. Furthermore, if there is significant encroachment of the floodplain from the roadway embankment, the modeler needs to request a more detailed survey of the embankment to make sure any model representation of the site depicts that encroachment. A modeler can use the Survey Request Form (Appendix B.2) to communicate to a survey crew and determine the appropriate level of detail that should be collected. Examples of survey information relating to bridges that may need to be collected are as follows:

- Breaklines depicting structure dimensions (e.g., arch details, low chord elevation, bridge deck height, etc.)
- Abutment geometry
- Roadway embankment geometry
- Piers (e.g., number of piers, shape and size, footing dimensions, etc.)
- Detailed survey depicting substructure and groundlines

- Geometry of scour holes (if present) around bridge foundation elements
- Wingwalls

B.1.2.2. Culverts

The unique culvert components that should be surveyed are listed below. The level of detail of each component should be determined by the modeler based on the project purpose and site features. For example, if a culvert at a project site is suspected to be undersized, the modeler should communicate to the survey crew that there may be a drop and a large scour hole at the culvert outlet that needs detailed points to be represented in the survey.

Additionally, the undersized culvert may promote deposition upstream, which would prompt the modeler to request more detail near the culvert inlet. The inlet and outlet elevations of a culvert should be surveyed, in addition to the elevation of the channel bed at both locations. All headwalls, wingwalls, and any other improved inlet features should be detailed using breaklines so that the structure hydraulics can be represented. Additionally, a modeler should notify a survey crew if the geometry of any existing baffles inside a culvert should be collected.

A modeler can then use a Survey Request Form (Appendix B.2) to communicate to a survey crew the appropriate level of detail that should be collected. Examples of some details that should be requested are as follows:

- Number of barrels
- Shape
- Length
- Inside diameter
- Span and rise
- Material type (concrete, CMP, HDPE, etc.)
- Inlet and outlet elevations
- Depth of sediment in barrel (at inlet and outlet)
- Headwalls
- Wingwalls
- Other unique culvert entrance and exit features
- Apron
- Outfall drop
- Baffles (dimensions, number, and spacing)
- Roadway embankment

B.1.2.3. Channel Bars or Islands

Representing channel bars or island features is important for evaluating hydraulic parameters at a project site with 2D modeling. To define channel bars or islands, breaklines defining the top of bank and toe of bank should be collected at a minimum. Additionally, enough points should be collected to represent the channel bar or island feature to meet project accuracy requirements.

B.1.2.4. Levees

A detailed depiction of levee components in a survey is important when evaluating floodplain extents for a project with 2D modeling. Misrepresentation of levee topography could affect the predicted model results, including floodplain extents. Breaklines depicting the toe and crown of a levee, top width, as well as its spatial extents, should be collected at a minimum. A modeler should illustrate to a survey crew where levee features are expected to be found on a project site.

B.1.2.5. Weirs

Weirs are features that need to be defined in a project survey to represent the hydraulics at a project site correctly. For example, a project may need to effectively tie-in to an existing set of weirs without adversely affecting upstream or downstream infrastructure or evaluate fish passage conditions. Breaklines detailing the weir notch dimensions, as well as points detailing pool topography, may be required to reflect project conditions properly.

By using the Survey Request Form (Appendix B.2), in conjunction with a figure, a modeler can direct a surveyor to the specific weir components that would include more attention. Below is an example list of items to identify to a survey crew:

- Weir dimensions and extents
- Downstream pool topography
- Breaklines depicting ground elevation directly upstream and downstream of weir
- Notch dimensions

B.1.2.6. Fish Ladder

If a project includes an existing fish ladder, it is important to communicate to a survey crew that the feature is present and its components need survey. A 2D model can represent the depth-averaged hydraulics through a fish ladder if the survey is collected so that breaklines and points define the components correctly.

B.1.2.7. Existing Scour Protection or Bank Stabilization Features

If the project site includes any existing scour protection or bank stabilization features (e.g., rock revetments, barbs, soil lifts, or large woody material), these features should be represented in a project survey. In some cases, a survey crew may not know what to look for so a modeler should provide a display (e.g., an aerial image) depicting the approximate location and extent of the feature(s). As previously mentioned, conducting a site visit prior to sending a survey request allows a modeler to note, with some accuracy, where the feature(s) can be found.

Additionally, the modeler should communicate with a survey crew whether the feature is expected to be inundated at the time of survey so that the survey crew can be prepared with the appropriate equipment. The full extents of protection should be collected. It is also suggested to request the survey crew to include notes and pictures of the scour protection indicating characteristics, such as the average size of rock (if the protection consists of rock).

B.1.2.8. Habitat Features

Habitat features may be necessary to include in a project survey for various reasons. For example, existing wetlands or large trees may need to be avoided during design and construction. In addition, existing large woody material in a channel, or fallen material along the banks, maybe influencing hydraulic conditions at the project site. A modeler should depict any existing habitat features of interest to a survey crew with a brief description and a display noting the location and approximate extents in the Survey Request Form (Appendix B.2).

By using the Survey Request Form, in conjunction with a figure, a modeler can direct a surveyor to the areas that would need more attention. Below is a list of items to identify for a survey crew that may be features of importance for a project site.

- Existing placed features such as logs with root wads or bioengineered slopes
- Notable existing trees along the bank
- Existing wood in the channel (location, length, diameter, etc.)
- Log jams and/or debris accumulations at or near a structure opening or near a channel bar or island feature
- Wetlands

B.1.2.9. Roadways

Surveying roadway components that are adjacent to water bodies and hydraulic structures is important for representing a project site in a 2D model. For example, a representation of a roadway surface would allow a modeler to assess the potential for overtopping of the roadway. The modeler should communicate to the survey crew the extents of survey required along a roadway. Using the Survey Request Form (Appendix B.2), in conjunction with a figure, a modeler can direct a surveyor to the areas that would need more attention. Below is a list of roadway elements that should be included in a project survey.

- Top of roadway elevation
- Guardrail or barrier
- Roadway width
- Detailed survey depicting roadway extents

B.1.2.10. Other Features

As each project site is unique, modelers should perform a desktop survey or site visit prior to completing a survey request. This enables the identification of any other features that need to be collected by the survey crew.

Using the Survey Request Form (Appendix B.2) in conjunction with a figure, a modeler can direct a surveyor to the areas that would need more attention. Additional items that may be necessary to include for a project site include:

- Utilities
- Ditches
- Stormwater outlets
- Upstream features (a dam or existing private water crossing)

- Property boundaries
- Private wells
- Buildings or other features within the floodplain
- Water diversion structures
- Irrigation channels
- Flood gates

B.2. EXAMPLE SURVEY REQUEST FORM

(Sample form is provided below)

SURVEY REQUEST FORM

Project Title: _____

Water Body: _____

Roadway:	Mile Post:
Latitude:	Longitude:
Preferred Datum:	Existing Data Sources Available:
Contact:	Contact Number:
Date of Request:	E-mail:

PROJECT EXTENTS

Upstream Channel Extents: (See Attached Display)	
Upstream Floodplain Extents: (See Attached Display)	Elevation
Downstream Channel Extents: (See Attached Display)	
Downstream Floodplain Extents: (See Attached Display)	Elevation

HYDRAULIC FEATURES

Feature Onsite	Feature	Important Feature Components	Notes
Circle One Yes No	Bridges	• Breaklines depicting structure dimensions (e.g., arch details, low chord elevation, bridge deck height, etc.); • Abutment geometry; • Roadway embankment geometry; • Piers (e.g., number of piers, shape and size, footing dimensions, etc.); • Detailed survey depicting substructure and groundlines; • Geometry of scour holes (if present) around bridge foundation elements; • Wingwalls; etc.	
Circle One Yes No	Culverts	• Number of barrels; • Shape; • Length; • Inside diameter; • Span and rise; • Material type (e.g. concrete, CMP, HDPE, etc.); • Inlet and outlet elevations; • Depth of sediment in barrel (at inlet and outlet); • Headwalls; • Wingwalls; • Other unique culvert entrance and exit features; • Apron; • Outfall drop; • Baffles (dimensions, number, and spacing); • Roadway embankment; etc.	
Circle One Yes No	Channel Bars or Islands	• Breaklines defining top of bank and toe of bank; • Sufficient shots to represent feature with full topo	

Feature Onsite	Feature	Important Feature Components	Notes
Circle One Yes No	Levees	• Breaklines depicting levee toe and crown of levee and spatial extents; • Top width; • Spatial extents of bank protection; etc.	
Circle One Yes No	Weirs	• Weir dimensions and extents; • Downstream pool topography; • Breaklines depicting ground elevation directly upstream and downstream of weir; • Notch dimensions; etc.	
Circle One Yes No	Fish Ladder	• All Dimension Details	
Circle One Yes No	Existing Scour Protection or Bank Stabilization Features	• Extents of protection; • Notes depicting type of protection (e.g., average size of rock riprap, dolosse, bioengineered slope, etc.); etc.	
Circle One Yes No	Habitat Features	• Existing placed features such as logs with root wads or bioengineered slopes; • Notable existing trees along the bank; • Existing wood in the channel (e.g., location, length, diameter, etc.); • Logjams and/or debris accumulations at or near a structure opening or near a channel bar or island feature; • Wetlands; etc.	

Feature Onsite	Feature	Important Feature Components	Notes
Circle One Yes No	Roadway	• Top of roadway and guardrail or barrier; • Roadway width; • Detailed survey depicting roadway extents; etc.	
Circle One Yes No	Other	• Utilities; • Ditches; • Stormwater discharges; • Upstream features (e.g., a dam or existing private water crossing); • Property boundaries; • Private wells; • Buildings or other features within the floodplain; • Water diversion structures; • Irrigation channels; • Flood gates; etc.	

APPENDIX C

2D MODEL REVIEW

C.1. REVIEW SUMMARY

Two-dimensional models significantly reduce the number of assumptions compared to using a 1D model. However, some assumptions are still necessary, various qualities of data are used, 2D models have inherent limitations, best practices should be followed, and potential for human error still exists. Any hydraulic modeling analysis workflow should include reviews.

Reviewing 2D models can take many forms. The model developer should review their model during development, before initial simulations, and assess the results of each simulation. In addition to the modeler being responsible for their work, it is advisable and often necessary, to implement additional QA/QC processes.

Model reviews can generally be categorized into two types: an informal review, and a formal review. Someone other than the modeler could conduct either type of review. Depending on project requirements the review could occur by staff within the same organization or as necessary, by a third party.

Model documentation in the form of metadata, review summary memo, or hydraulic modeling report should be provided with the model files to assist with the review.

C.2. SAMPLE REVIEW SUMMARY MEMO OUTLINE

The following outline is an example of what to include in a summary memo to be provided to a reviewer. Modelers should adjust this outline to fit specific project needs and to be able to provide sufficient information to assist with the review.

1. **INTRODUCTION**
   1.1. Project Description, purpose and location

2. **HYDROLOGIC ANALYSIS**
   2.1. Results of hydrologic analysis

3. **MODEL DEVELOPMENT**
   3.1. Topographic and Bathymetric Data Sources
       3.1.1. Coordinate systems, vertical datum, and accuracy
       3.1.2. Summary of final merged surface
   3.2. Mesh Generation
       3.2.1. Downstream and Upstream Limits (and why chosen)
       3.2.2. General Statistics (area covered, number of elements)
       3.2.3. Mesh quality
   3.3. Boundary Conditions
       3.3.1. Upstream/downstream boundary configuration and values
       3.3.2. Internal Boundary Conditions
   3.4. Roughness Assignments
       3.4.1. Materials and Manning's n values used
   3.5. Run Control Parameters

4. **HYDRAULIC ANALYSIS RESULTS**
   4.1. Continuity check results
   4.2. Calibration process if applicable
   4.3. Sensitivity analysis if applicable

C.3. INFORMAL

An informal review focuses on the reasonability of the model and results. It is intended to be conducted in a short amount of time compared to a formal review. This type of review should answer the following questions:

- Does the model generally represent the project site?
- Do the results appear reasonable?

C.4. FORMAL

A formal review is more thorough than an informal review. This type of review seeks to confirm the reasonability of all items that might impact the quality of the model, including:

- Terrain data
- Mesh geometry
- Material assignments and values
- Boundary conditions
- Representation of hydraulic structures
- Continuity and stability of model
- Calibration if applicable

Checklists

A sample of each review checklist is provided below. Excel files of each review checklist are available on FHWA's hydraulics website. When performing reviews, it is helpful, and advised, to include screenshots of items requiring action. Instructions for including notes and screenshots are included in both review checklists.

Informal checklist sample

2-D Hydraulic Model Review Checklist

Informal Review

Project: _____   Reviewer: _____

River: _____   Modeler: _____

Project Purpose: _____   Date: _____

Project File Name: _____

Item	Comment	Action Needed (blank = none)	Response to Comment/Resolution	Screen Shot
Mesh				
Are the number of elements reasonable?				X
Does the upstream mesh limit appear sufficient?				
Does the downstream mesh limit appear sufficient?				
Are the lateral extents sufficient?				
Hydraulic Structures				
Bridge				
Do bridges appear to be represented correctly?				
Do model results appear reasonable at bridges?				
Culvert				
Do culverts appear to be represented correctly?				
Do model results appear reasonable at culverts?				
Other Structures				
Do model results appear reasonable at other structures?				
Hydraulic Analysis				
Are simulation settings reasonable				
Do results appear to reach steady state?				
Do results contain any oddity's				
General Comments				

C.3

Formal checklist sample

2-D Hydraulic Model Review Checklist
Formal Review

Project: _____   Reviewer: _____
River: _____   Modeler: _____
Project Purpose: _____   Date: _____
Project File Name: _____

Item	Comment	Action Needed (blank = none)	Response to Comment/Resolution	Screen Shot
Model/Background Data				
Version of SMS/SRH-2D				
Project Vertical Datum				
Project Horizontal Datum				
Documentation of techniques and procedures				
Metadata included in model files				
Topography				
Source/Date				
Stated Accuracy				
Datums verified				
Bathymetry				
Source/Date				
Datums verified				
Additional Survey				
Source/Date				
Datums verified				
Bridge/Culvert/Structure Data				
Source/Date				
Datums verified				
Terrain Data review				
Data consistency				X
Does the final surface accurately represent the site (are hydraulic controls represented)				
Confirm breaklines used where necessary				
Mesh				
Mesh stats: number of elements, length, floodplain widths				
Is the number and size of mesh elements reasonable?				
Is the upstream mesh limit sufficient?				
Is the downstream mesh limit sufficient?				
Are the lateral extents sufficient				
Are key project features correctly represented?				
Is mesh quality sufficient?				
Materials				
Source of material types (imagery)				
Are material types correctly assigned				
Are appropriate n values used				

C.4

Boundary Conditions				
Steady or unsteady simulation?				
Upstream Boundary - Verify correct inflow(s) amount and type				
Downstream Boundary - Verify correct stage and type				
Are boundary conditions applied to mesh correctly				
Hydraulic Structures				
How many structures are represented? What types?				
Bridge				
Is bridge geometry correct				
Are pier locations correct				
Are piers modeled correctly				
Is pressure flow accounted for correctly				
Culvert				
Does the mesh align with the culvert limits?				
Are the BC arcs placed correctly?				
Is culvert correctly represented				
Obstructions				
Are obstructions used in the model? If so, are they applied correctly with correct values?				
Other Structures				
If other structures are included, are they correctly represented				
Hydraulic Analysis				
Model Controls and Simulations				
Are simulation settings reasonable				
Are simulations labeled correctly and include the correct components				
Model Results				
Confirm model stability at monitoring points				
Confirm continuity at monitoring lines				
Confirm stable results through the domain				
Do results contain any oddity's (check WSE, Velocity, Depth, Shear, Froude #)				
Calibration				
Was the model calibrated, if so, do the model results match the calibrated data				
If no calibration was performed, was a sensitivity analyses performed, if so what was determined				
General Comments				

C.5

APPENDIX D

2D MODEL REPORT OUTLINE

TWO-DIMENSIONAL HYDRAULIC MODELING REPORT OUTLINE

i.   Table of Contents
ii.   List of Figures
iii.   List of Tables

1. INTRODUCTION
1.1. Project Description
     1.1.1. Sponsor/authorization
     1.1.2. Purpose of Study
     1.1.3. Project Location
     1.1.4. Previous Relevant Studies

2. HYDROLOGIC ANALYSIS
2.1. Watershed Description
2.2. Nature of Flood Risk
     2.2.1. Flood History
     2.2.2. Types of Events
2.3. Previous Hydrologic Studies in Watershed
     2.3.1. FEMA, USACE, USGS, DOT, or others
2.4. Flood Frequency Relationship used and why
- per FEMA (if any previous study)
- Gage Data (if relevant gage data exists)
- Regional Regression Equations
- Rainfall-Runoff Analysis (if available or undertaken for study)

3. MODEL DEVELOPMENT
3.1. Topographic and Bathymetric Data
     3.1.1. Sources
     3.1.2. Vertical Datum (should be NAVD 88 except in special circumstances)
     3.1.3. Horizontal Coordinate System
     3.1.4. Key Topographic and Structural Hydraulic Controls
3.2. Mesh Generation Existing Conditions
     3.2.1. Downstream and Upstream Limits (and why chosen)
     3.2.2. General Statistics (area covered, number of elements)
     3.2.3. Areas of High Density (and why)
     3.2.4. Mesh quality
     3.2.5. Representation of Terrain
     3.2.6. Model Calibration and Sensitivity
3.3. Boundary Conditions Existing Conditions
     3.3.1. Upstream Boundaries Configuration
     3.3.2. Discharge Values (relate to Hydrologic Analysis)
     3.3.3. Downstream Boundary Value Assignments
     3.3.4. Internal Boundary Conditions
3.4. Roughness Assignments Existing Conditions
     3.4.1. Designation of Materials (how many and why)
     3.4.2. Assigned Manning Coefficient Values
3.5. Run Control Parameters
3.6. Proposed Conditions
     3.6.1. Alternate 1 (modifications from existing conditions)
     3.6.2. Alternate 2 (etc.)

4. HYDRAULIC ANALYSIS RESULTS FOR EXISTING CONDITIONS
    4.1. Water Surface Results
    4.2. Velocity Results
    4.3. Depth Results
    4.4. Discharge Distribution
        4.4.1.   Check for model continuity
        4.4.2.   Determine flow distribution at flow splits, multiple openings, etc.
    4.5. Shear Stress Results if relevant
    4.6. Other Project Specific Results Discussions

5. HYDRAULIC ANALYSIS RESULTS FOR PROPOSED CONDITIONS
    5.1. Water Surface Results
    5.2. Velocity Results
    5.3. Depth Results
    5.4. Discharge Distribution
        5.4.1.   Check for model continuity
        5.4.2.   Determine flow distribution at flow splits, multiple openings, etc.
    5.5. Shear Stress Results if relevant
    5.6. Relevant Differences from Existing Conditions

6. HYDRAULIC DESIGN CONSIDERATIONS
    6.1. Bridge Freeboard and Hydraulic Capacity
    6.2. Freeboard for Road Surfaces
    6.3. Culvert Hydraulic Parameters
    6.4. Avoiding/Mitigating Unacceptable Water Surface Impacts
    6.5. Regulatory Requirements

7. STREAM STABILITY AND SCOUR EVALUATION

8. DESIGN OF SCOUR AND STREAM INSTABILITY COUNTERMEASURES

9. OTHER PROJECT SPECIFIC TOPICS
    • AOP Studies
    • Flood Mitigation
    • Sediment Transport

10. CONCLUSION

APPENDIX (as needed)

<u>Figures to include</u>
(Some figures can be included in an appendix if necessary)

Chapter 1
　　1.　Project vicinity/location map
Chapter 2
　　1.　Watershed map
　　2.　Flood-frequency curve from each method examined
　　3.　Adopted flood-frequency curve for this study
Chapter 3
　　1.　Plot of terrain surface
　　2.　Plot of full mesh
　　3.　Plots of specific high-density areas
　　4.　Mesh quality plot
　　5.　Elevation contour plots demonstrating good representation of terrain
　　6.　Annotated plot showing all boundary conditions
　　7.　Plot of materials coverage with legend
　　8.　Plot of alternative materials coverage if different from existing
Chapter 4
　　1.　WSEL contours
　　2.　Profile along river
　　3.　Profiles along overbanks or floodplain edges
　　4.　Velocity magnitude contours and vectors, overall and zoomed in to areas of interest
　　5.　Depth contour plots
　　6.　Plot showing discharge observation lines
　　7.　Shear stress contour plots
Chapter 5
　　1.　WSEL contours
　　2.　Profile along river
　　3.　Profiles along overbanks or floodplain edges
　　4.　Velocity magnitude contours and vectors, overall and zoomed in to areas of interest
　　5.　Depth contour plots
　　6.　Plot showing discharge observation lines
　　7.　Shear stress contour plots
　　8.　Profile and contour plots showing impacts of proposed conditions
Chapter 7
　　1.　1% and 0.2% AEP event scour profile plots (include max scour event plot if different)

<u>Tables to include</u>

Chapter 2
　　1.　Flood-frequency table showing values from each method examined
　　2.　Adopted flood-frequency table
Chapter 3
　　1.　Table of Manning coefficients per material type
Chapter 4
　　1.　Table of discharges
Chapter 5
　　1.　Table of discharges
Chapter 7
　　1.　Scour analysis results