US Army Corps
of Engineers
Hydrologic Engineering Center

Modeler Application Guidance for Steady vs Unsteady, and 1D vs 2D vs 3D Hydraulic Modeling

August 2020

TD-41

REPORT DOCUMENTATION PAGE	*Form Approved OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)*	**2. REPORT TYPE**	**3. DATES COVERED** *(From - To)*
August 2020	Training Document	

4. TITLE AND SUBTITLE	**5a. CONTRACT NUMBER**
Modeler Application Guidance for Steady versus Unsteady, and 1D versus 2D versus 3D Hydraulic Modeling	**5b. GRANT NUMBER**
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	**5d. PROJECT NUMBER**
Gary Brunner, P.E., D.WRE., M. ASCE Gaurav Savant, Ph.D., P.E. Ronald E. Heath	**5e. TASK NUMBER**
	5F. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	**8. PERFORMING ORGANIZATION REPORT NUMBER**
U.S. Army Corps of Engineers Institute for Water Resources Hydrologic Engineering Center (CEIWR-HEC) 609 Second Street Davis, CA 95616-4687 US Army Corps of Engineers Engineer Research and Development Center Coastal and Hydraulic Laboratory 3909 Halls Ferry Road Vicksburg, MS 39180	TD-41

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	**10. SPONSOR/ MONITOR'S ACRONYM(S)**
	11. SPONSOR/ MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION / AVAILABILITY STATEMENT
Approved for Public Release. Distribution Unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT
All models, numerical or scale-physical, are simplified representations of the real world (prototype). Fortunately, there are numerous practical engineering problems for which simplified numerical models of the prototype are sufficient to provide usable descriptions of system behavior. The challenge for the modeler is to select an appropriate model to solve their particular engineering problem while recognizing that the model is not a perfect representation of the prototype. Selection of a model begins with developing an understanding of which aspects of the complex, real-world system are most important to the engineering problem being addressed. The purpose of this document is to provide entry to mid-level hydraulic engineer's with guidance on when to use Unsteady Flow modeling instead of Steady flow modeling; and how to select between one-dimensional (1D), two-dimensional (2D), or three-dimensional (3D) modeling for a given problem.

15. SUBJECT TERMS
water surface profiles, river hydraulics, steady and unsteady flow, one-dimensional, two-dimensional, and three-dimensional hydrodynamics, computer program, numerical model, HEC-RAS, AdH.

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			
U	U	U	UU	114	**19b. TELEPHONE NUMBER**

Modeler Application Guidance for Steady vs Unsteady, and 1D vs 2D vs 3D Hydraulic Modeling

August 2020

US Army Corps of Engineers
Institute for Water Resources
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616

(530) 756-1104
(530) 756-8250 FAX
www.hec.usace.army.mil

US Army Corps of Engineers
Engineer Research and Development Center
Coastal and Hydraulic Laboratory
3909 Halls Ferry Road
Vicksburg, MS 39180

(601) 634-2502
https://www.erdc.usace.army.mil
https://chl.erdc.dren.mil

TD-41

Table of Contents

List of Tables iii
List of Figures v
Abbreviations ix

Chapter 1 1-1
Introduction 1-1
Acknowledgements 1-2

Chapter 2 2-1
Knowledge of the Hydraulic System and Purpose of the Hydraulic Model 2-1
Purpose of the Model 2-1
Knowledge of the River System 2-2
Sources and Accuracy of the Data 2-4
Duration of the Events to be Modeled 2-5
Required Model Outputs 2-6
Experience of the Modeler 2-7

Chapter 3 3-1
Data Requirements 3-1
Terrain Data 3-1
Roughness Coefficients 3-2
Underground Drainage Systems 3-3
Hydraulic Structures 3-4
Hydrology 3-5
Boundary Conditions 3-5
Calibration Data 3-5

Chapter 4 4-1
Model Output/Results 4-1
Varying Levels of Detail in Hydraulic Outputs 4-1
Expected/Desired Level of Accuracy 4-6

Chapter 5 5-1
Steady Flow vs Unsteady Flow Modeling 5-1
Definitions 5-1
Steady Flow Assumptions 5-2
Hydrologic vs Hydraulic Routing 5-4
Computational Differences 5-4
Calibration Strategies 5-11
Steady Flow Modeling Limitations 5-12
Unsteady Flow Modeling Limitations 5-13

Chapter 6 6-1
One-Dimensional vs Two-Dimensional Modeling 6-1
Definitions 6-1
One and Two-Dimensional Equations 6-2
Computational Differences 6-5
Application Examples 6-12

Model Development 6-19
Model Calibration 6-25
Time and Cost Issues 6-28
Modeler Knowledge, Skills, and Abilities 6-29
Summary of 1D and 2D Modeling Advantages and Disadvantages 6-29

Chapter 7 7-1
Two-Dimensional vs Three-Dimensional Modeling 7-1
Two and Three-Dimensional Equations 7-1
Application Examples 7-3
Model Development 7-16
Model Calibration 7-21
Time and Cost Issues 7-22
Modeler Knowledge, Skills, and Abilities 7-23
Summary of 2D and 3D Modeling Advantages and Disadvantages 7-23

Chapter 8 8-1
Physical Hydraulic Models 8-1

Chapter 9 9-1
Summary 9-1

References 1

List of Tables

Table 4-1. Hydraulic Model Outputs and 1D, 2D, and 3D Level of Detail. 4-1
Table 7-1. Suggested Turbulence Closure Schemes 7-20
Table 9-1. Recommended modeling for various commonly modeled systems........................ 9-2

List of Figures

Figure 2-1. Multiple flow paths for water moving inside of a leveed system after a breach. .. 2-4
Figure 2-2. Example of Detailed LIDAR and channel data (left) versus 10m DEM data (Right). 2-5

Figure 3-1. Terrain model without under water channel data. 3-2
Figure 3-2. Terrain model with channel bathymetry burned into terrain model. 3-2
Figure 3-3. Example of land use and user defined polygons to define roughness for a 2D model. 3-3
Figure 3-4. Detailed terrain and 2D modeling mesh of the 17th St. outfall canal in New Orleans, LA. 3-4

Figure 4-1. Example 1D versus 2D Water Surface Elevation Plot. 4-3
Figure 4-2. One-dimensional (1D) velocity plot at an example cross section. 4-4
Figure 4-3. Two-dimensional (2D) velocity plot at an example cross section. 4-4
Figure 4-4. Example 2D and 3D velocity plots through gate openings. 4-5
Figure 4-5. One-dimensional (1D) model results for an interior area with a levee breach. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth). 4-5
Figure 4-6. Two-dimensional (2D) model results for an interior area with a levee breach. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth). 4-6

Figure 5-1. Steep stream (slope = 5 ft/mile) with profiles computed using maximum flows and instantaneous flows. 5-3
Figure 5-2. Flat stream (slope = 0.5 ft/mile) with profiles computed using maximum flows and instantaneous flows. 5-3
Figure 5-3. Forces acting on a body of water from cross section 2 to cross section 1. 5-5
Figure 5-4. Example family of rating curves pre-computed for a bridge. 5-8
Figure 5-5. Calibrated Steady Flow Model and Unsteady flow model with and without contraction and expansion losses added. 5-9
Figure 5-6. Example layout of ineffective flow areas (black diagonal-line-filled polygons) for 1D modeling. 5-10
Figure 5-7. Hydrograph going into and out of a river reach with ineffective flow areas acting as storage. 5-11

Figure 6-1. Definition of Symbols used in the 1D and 2D equations of motion. 6-3
Figure 6-2. Example of a leveed system breach with water going in many directions. 6-13
Figure 6-3. Example Dambreak that goes out into an extremely flat area and spreads out. Water depths shown in shades of blue (dark blue indicates greater water depth). 6-13
Figure 6-4. Lower Columbia River Bay with water depths shown in shades of blue. 6-14
Figure 6-5. Example of super elevation of the water surface around a sharp bend. 6-15
Figure 6-6. Detailed 2D-Laterally Averaged model of vertical velocity gradients. The directions shown are x and z. 6-15

List of Figures

Figure 6-7. Detailed 2D model of flow going around piers from a railroad station platform. ...6-16
Figure 6-8. Example of a highly one-dimensional flowing river system (Allegheny - Monongahela Rivers, confluence at Pittsburgh, PA). Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth).........6-17
Figure 6-9. Example 1D/2D model of the Truckee River near Reno, NV. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth). 6-18
Figure 6-10. Example 2D mesh with a detailed mesh of the main channel, with grids aligned to the flow. Green to red color indicates the terrain (low to high elevation). 6-21
Figure 6-11. Example 2D mesh with a detailed mesh of increased resolution around structures. Green to red color indicates the terrain (low to high elevation).........6-22
Figure 6-12. Example velocity output for a detailed 2D model of a bridge (velocity overlays terrain where green to red color indicates low to high elevation).........6-24
Figure 6-13. Example of a Calibrated 1D model for the Lower Columbia River. 6-27

Figure 7-1. Definition of symbols used in 3D equations. 7-3
Figure 7-2. Deep channel surrounded by shallow areas, red indicates deeper and green shallower. 7-4
Figure 7-3. Salinity stratification and velocity differences in an estuary. 7-4
Figure 7-4. Temperature stratification in a lake.........7-5
Figure 7-5. Computed depth-averaged velocity field in the Mississippi River with vorticity transport.........7-6
Figure 7-6. Change in computed velocity produced by vorticity transport method. 7-7
Figure 7-7. Helical flow in a river bend.........7-7
Figure 7-8. Velocity difference between surface and bottom in a river bend.........7-8
Figure 7-9. Averaged velocity comparison between 2D (depth averaged) and, 3D-NHMP (unpressurized) models for a straight reach of a river with nine equally sized gates. Colors scaled to illustrate patterns, not exact values. 7-10
Figure 7-10. Depth averaged velocity comparison between 2D, and 3D-NHMP (pressurized) models for a straight reach of a river with nine gates where the central gate intrudes 0.5 meters into the water column. Colors scaled to illustrate patterns, not exact values. 7-11
Figure 7-11. Bathymetry and 2D-velocity for a sloped spillway that must convey a Probable Maximum Flood (PMF) of 6,179 m^3/sec. 7-12
Figure 7-12. Sloped spillway (which must convey a PMF of 6,179 m^3/sec), 3D velocity (colors represent velocity) and water surface (thickness represents water surface) results. 7-13
Figure 7-13. Bathymetry, and 2D-velocity and water surface profile for the sloped spillway (which must convey a PMF of 6,179 m^3/sec). 7-13
Figure 7-14. Bathymetry for a stepped spillway which must pass a PMF of approximately 3,145 m^3/s.........7-14
Figure 7-15. Results for the 3D-NHMP simulated hydraulics for a stepped spillway (which must pass a PMF of approximately 3,145 m^3/s).........7-14

List of Figures

Figure 7-16. Results for the 2D simulated hydraulics for a stepped spillway (which must pass a PMF of approximately 3,145 m^3/s) for different Manning's roughness (n). Thickness indicates depth. 7-15
Figure 7-17. Meshing Strategies for 3D. 7-17
Figure 7-18. Horizontal meshing for 3D models, green to red color indicates the bathymetric features (deeper to shallower). 7-18
Figure 7-19. Example of Arbitrary-Lagrangian Eulerian (ALE) vertical meshing for 3D models. 7-19

Abbreviations

1D	one-dimensional
2D	two-dimensional
3D	three-dimensional
3D-NH	3D-Non Hydrostatic
3D-NHMP	3D-Non Hydrostatic Multi Phase
AdH	Adaptive Hydraulics
ALE	Arbitrary-Lagrangian Eulerian
CEIWR	USACE Institute for Water Resources
CELRH	USACE Huntington District
cfs	cubic feet per second (ft^3/sec)
cms	cubic meters per second (m^3/sec)
CPU	central processing unit
DEM	digital elevation model
DTM	digital terrain model
e.g.,	APA Style Latin Abbreviation for "for example"
ERDC	USACE Engineer Research and Development Center
ERDC-CHL	USACE-ERDC Coastal and Hydraulics Laboratory
ft	feet
GPU	graphics processor units
HEC	CEIWR Hydrologic Engineering Center
HEC-HMS	HEC Hydrologic Modeling System software
HEC-RAS	HEC River Analysis System software
HPC	high performance computers
i.e.,	APA Style Latin Abbreviation for "that is"
LIDAR	Light Detection and Ranging
kg	kilogram
m	meter
NOAA	National Oceanic and Atmospheric Administration
NWS	National Weather Service
PMF	probable maximum flood

Abbreviations

ppt	parts per thousand
RAM	random access memory
sec	second
USACE	U.S. Army Corps of Engineers
WS	water surface
WSE	water surface elevation

Chapter 1

Introduction

All models, numerical or scale-physical, are simplified representations of the real world (prototype). When properly applied, most modern numerical models will provide reasonably accurate solutions of basic conservation equations (conservation of energy, mass, and momentum). However, characterization of complex, hydraulic systems generally requires parameterizations, often purely empirical, to describe physical processes that cannot be directly determined from the solution of the basic conservation equations. The classic example of an empirical parameterization in hydraulic modeling is the use of Manning's n-value to define hydraulic roughness.

Fortunately, there are numerous practical engineering problems for which simplified numerical models of the prototype are sufficient to provide usable descriptions of system behavior. The challenge for the modeler is to select an appropriate model to solve their particular engineering problem while recognizing that the model is not a perfect representation of the prototype. Selection of a model begins with developing an understanding of which aspects of the complex, real-world system are most important to the engineering problem being addressed.

The purpose of this document is to provide entry to mid-level hydraulic engineer's with guidance on when to use Unsteady Flow modeling instead of Steady flow modeling; and how to select between one-dimensional (1D), two-dimensional (2D), or three-dimensional (3D) modeling for a given problem. As this document is meant to be a practical guide to hydraulic model applications, detailed theoretical derivations/discussions of the 1D, 2D, and 3D equations will not be presented.

This document will cover the following:

- Knowledge of the River, or other hydraulic system and Purpose of the Hydraulic Modeling.
- Data requirements for steady vs unsteady, and 1D vs 2D vs 3D models.
- Output/results provided by 1D, 2D and 3D models.
- Steady flow vs unsteady flow modeling (1D and 2D).
- 1D vs 2D Modeling
- 2D vs 3D Modeling.

Software used in the development of this document include:

- HEC-RAS: used to for 1D and 2D modeling examples.
- AdH: used for 2D, 3D, and 3D-Non Hydrostatic modeling examples.
- OpenFOAM: used for 3D-Non Hydrostatic Multi Phase modeling examples.
- PROTEUS: used for 3D-Non Hydrostatic Multi Phase modeling examples.

Acknowledgements

Chapters 1 through 6 of this document was written by Mr. Gary W. Brunner (CEIWR-HEC). Chapters 7, and 9 of this document was written by Dr. Gaurav Savant (USACE-ERDC). Chapter 8 was written by Mr. Ronald E. Heath (USACE-ERDC-CHL). All three authors collaborated on the initial review of the draft chapters. OpenFOAM results were provided by Mr. Brian Hall, and Mr. Nicholas Koutsunis (both from CELRH). PROTEUS results were provided by Dr. Christopher E. Kees (USACE-ERDC) and Mr. Jason H. Collins (USACE-MVN).

Chapter 2

Knowledge of the Hydraulic System and Purpose of the Hydraulic Model

To answer the questions of Steady vs Unsteady Flow, and 1D vs 2D vs 3D modeling approaches, the modeler must have knowledge of the hydraulic system to be modeled, as well as a clear understanding of the purpose of the model to be developed. Each system is unique and will have site specific information that must be considered in order to make an appropriate modeling choice. The following is a list of some of the things that should typically be consider before trying to make a modeling approach decision.

Purpose of the Model

The purpose of the model, and the expected level of accuracy required, can significantly dictate the modeling approach and required level of accuracy of the source data. Hydraulic models are developed for all kinds of purposes, such as: developing a model to produce rough answers quickly; a detailed planning study used to evaluate study alternatives; a design study in which the model will be directly used to design a structure; real time modeling and mapping, consequence mapping for dam or levee failure, etc.

Models that need to be developed quickly to provide rough answers will tend to be 1D or 2D models that are not very detailed. These types of models may be run in a steady flow or unsteady flow mode, in order to compute water surfaces and velocities that are approximate. Sometimes rough models are developed for emergency operations when no existing model exists for an area experiencing a major event. For a case such as this, the development of a simple 2D model is sometimes faster than the development of a 1D model, especially if a digital terrain model already exists and is detailed enough to capture important hydraulic features. In other words, to layout a 2D flow area (create a simple mesh, and attach some boundary conditions (flow and/or precipitation) to the mesh), is very easy. However, this type of 2D model is not a detailed model, and should not be viewed as one just because it is solving the 2D flow equations over a computational mesh. A detailed 2D model is generally just as much effort as a detailed 1D model, due to the fact that the user needs to spend a significant amount of time creating an appropriate computational mesh, defining land surface roughness for the entire spatial area, and calibrating the model.

Detailed planning studies are generally done with either 1D, 2D, or combined 1D/2D models that are computed in steady or unsteady flow mode. For a planning level of study, the specifics of the river system and floodplain (as described above in the "Knowledge of the River System" section), as well as the required model outputs for the study, will dictate the choice of model. Generally, 3D models are not used in planning studies unless the entire study is devoted to the design/analysis of a single hydraulic structure.

For detailed design level studies, it is common to use a 2D or even a 1D model as a preliminary screening tool (i.e., to reduce the number of design alternatives to model in detail). However, 3D models and physical models are more often used to design hydraulic structures, such as: spillways, weirs, gate openings, stilling basins/energy dissipaters; flow intake structures; pressurized pipe systems; complex stream junctions; fish passages; piers and abutments for bridges; river groins/training structures; complex river bends; and the like. Further, 3D models can also be used to perform detailed analysis of existing structures.

Real time river forecasting is another common area requiring hydraulic models. Due to the fact that real time forecasting requires models that run quickly, most forecasting systems use 1D and or combined 1D/2D models. However, if the forecast area is not too large, then more detailed 2D models can also be applied, as the computational requirements may not be that great. The need for quick answers that are reasonably accurate is the main driving force. Additionally, real time forecasting systems generally only need hydraulic models to produce water surfaces, flow rates, and inundation maps, and not detailed 2D or 3D velocity distributions.

Transport models, such as water quality and sedimentation models, can be sensitive to the accuracy of computed hydraulics. Hydraulic models used in conjunction with transport models may require additional validation to accurately resolve transport phenomena. For example, a specific combination of channel and floodplain roughness coefficients in a 1D model may reproduce observed river stages. However, if the modeled variation of channel velocity with stage differs from the prototype, sediment transport computations may produce excessive scour or deposition in the channel. In this case, additional adjustments to the flow distribution between the channel and floodplain may be required to obtain reasonable results from the sedimentation model.

Knowledge of the River System

The modeler must be aware of the physical description of the channels, floodplain areas, bridges/culverts, dams, levees, roads, other hydraulic structures that the model will be applied to, before making a modeling approach choice. Some typical questions to answer are:

What is the size/ length of the systems to be modeled? Is the extent of the system to be modeled 1 mile, 10, 50, 100, 500, or 1,000 miles? The length of the system to be modeled may dictate what level of modeling can be used. For example, one application is a forecast model for a very larger system. The modeler should not develop a detailed 2D model of 100 miles of river system, and expect that model to run in a reasonable amount of time on a desktop workstation. Therefore, extremely large systems may need to be modeled in 1D, or combined 1D and 2D, in order to have a model that will run in a reasonable amount of computational time. In this instance, a 2D model may be desirable to inform the 1D model that needs to run more quickly to support a forecast model.

What is the complexity of the system to be modeled? There are many factors that are part of the complexity question. Is the system hydraulically steep, or does it have steep portions of the system? Hydraulically steep systems can be more difficult to model than flat river systems. In a hydraulically steep system there are higher velocities, more rapid changes in depth, area, and velocity, which occur over very short distances. There may also be supercritical flow, and flow transitions from sub to supercritical and supercritical to subcritical. These conditions may make computation of stable solutions to the conservation equations challenging and special techniques may be required in order to solve the equations.

Another factor that is part of the complexity question is the number and type of hydraulic structures in the system. Typical questions are:

- Are there many bridges that impact the water surface elevations during higher flows?
- Are there severely skewed bridges or bridges in very wide floodplains with variable overtopping water surface elevations?
- Are there non-pressure flow bridges with complex piers or abutments?
- Are there dams and weirs to be modeled?
- Are there dam or levee breaches that need to be simulated?
- Are there gated structures that require unique ways in which the gates must be operated during events?

If there are numerous hydraulic structures in the system that need to be modeled, then a modeling approach should be selected that can accurately capture the most important aspects of how those structures affect the hydraulics of the system. The modeling approach will also depend on what level of detail is needed to model a particular structure. If modeling a larger system with many structures is required, it may not be possible to have a single model that is very detailed in the approach to modeling each individual structure (i.e., it will not be possible to have each structure modeled as a detailed 3D or even 2D structure). However, that may not be that important. If the goal of the model is to predict water surface elevations and flow rates within the system, then detailed knowledge of the 2D/3D velocity distribution through individual structures is not necessarily important.

Is the flow path of the water generally known for the full range of events? Understanding the flow path of the water for the full range of events is very important in the model decision making process. If the flow path of the water is well defined for the full range of events, then a 1D modeling approach can be used, as long as it is valid for the other aspects of the model. However, if the flow path of the water is unknown for some of the events to be modeled, or the water may split and go into several directions (i.e., water going over or through a levee may spread out in several directions once it enters the interior area, Figure 2-1), then using a 2D modeling approach for those areas is more appropriate than 1D modeling. Additionally, if the flow path of the water can change significantly during the event, 2D modeling approaches can handle this, whereas 1D modeling approaches cannot.

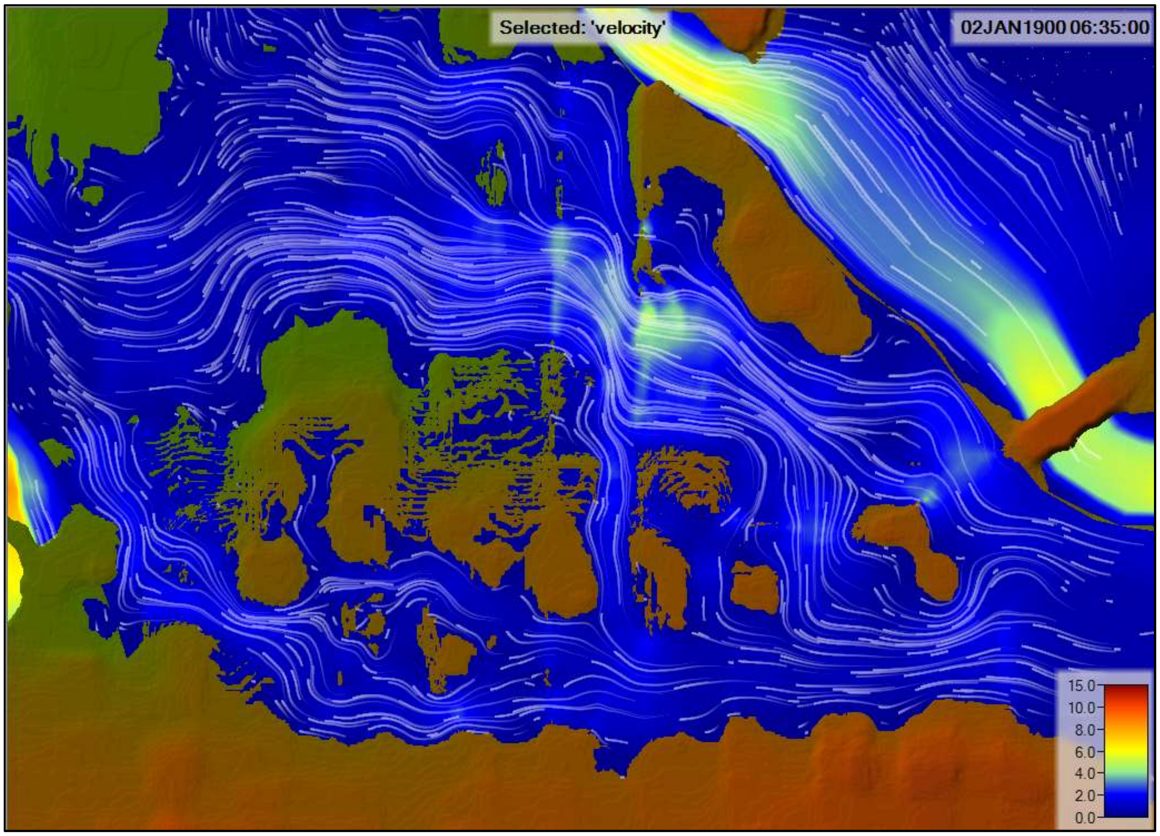

Figure 2-1. Multiple flow paths for water moving inside of a leveed system after a breach.

Are there unique aspects of the system that will significantly affect the computed results? When studying the system to be analyzed, the modeler should consider unique aspects of the system that are important to accurately depict the movement of the water and the resulting water surface elevations/flood inundation boundaries. Some examples of unique system features that will significantly affect the results of the model are: the system is tidally influenced, such that ocean tides have a significant impact on the water surface elevations; wind speed and direction has historically affected the water surface elevations; the river is affected by floating ice or ice jams; there tends to be debris issues during flood, and the debris tends to pile up at hydraulic structures (bridges, culverts, dams, etc.); there are levee systems that may be overtopped or breached, where interior flow routing needs to be addressed; and there are unique hydraulic structures that require specialized modeling or gate operations.

Sources and Accuracy of the Data

The source and level of accuracy of the data being used to develop the model is very important to the decision of the modeling approach and accuracy of the model results. Specifically, the level of detail and accuracy of the terrain data, bathymetric data, cross section data, levee information, and hydraulic structure data, is important in deciding how detailed a model can be developed. For example, if detailed terrain data and bathymetric data does not exist (i.e., only 10 meter DEM is available, but there are surveyed cross sections, see example in Figure 2-2), then the perceived increase in accuracy of using a 2D model over a 1D model may not actually be realized. The type of data, and its level of accuracy, will influence the quality of the modeling choice.

Additionally, the level of accuracy of the hydrology/boundary conditions used to drive the model must also be considered in the model selection process. If only estimates of peak flows are available, and no knowledge of the full hydrographs at the external and internal locations of the model, then unsteady flow modeling may not be possible.

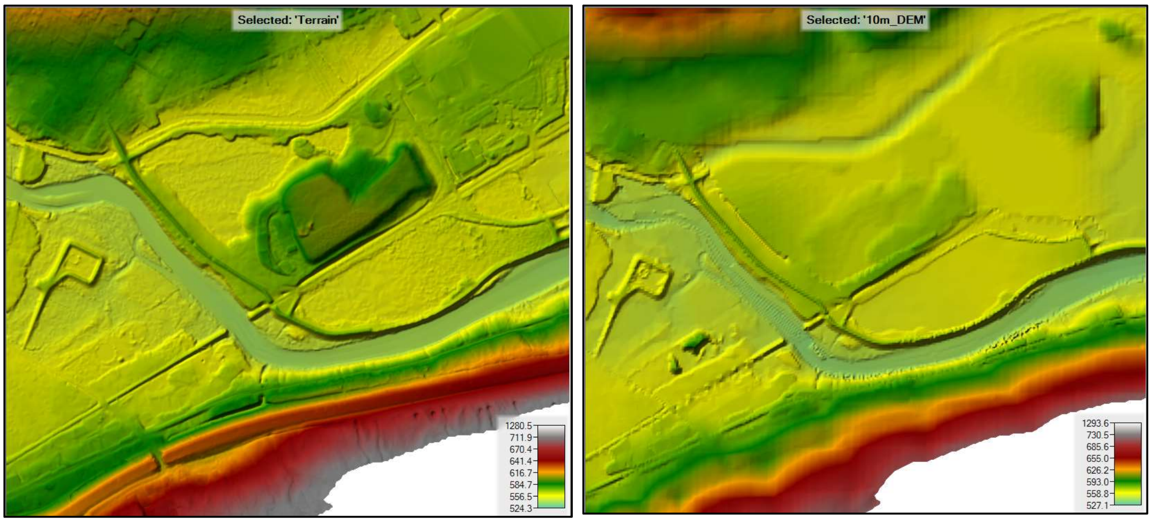

Figure 2-2. Example of Detailed LIDAR and channel data (left) versus 10m DEM data (Right).

Duration of the Events to be Modeled

Event types range from: a steady flow rate of a specific magnitude; a normal rainfall runoff type of event; flash floods; dam and/or levee breaching; flow releases from a hydraulic structures; etc. The duration of an event depends on the size of the watershed/river system being modeled, as well as study purpose. Some study purposes may only require the modeling of peak flows for a range of events. Indeed, 1D, 2D, and 3D models can all be used in a steady flow mode. However, 1D models are generally used to model long expanses of river systems based on peak flows derived from hydrologic models or observed data. In general, 2D and 3D models are used in a "steady flow mode" (running unsteady flow using a constant flow and/or stage hydrograph boundary conditions) for short reaches of river, or for the design and analysis of hydraulic structures.

For unsteady flow modeling, the duration of the events can have an impact on the type of hydraulic modeling approach. If the events being modeled are shorter in duration (i.e., 1 to 3 days, or less than a week), then 1D, 2D, or even 3D models may still be a viable choice, as long as the area being modeled is small. As the model domain becomes larger, then 3D models may no longer be a viable choice for locations with events longer than one week. As the event length goes from a few weeks to months, if the river system is of a significant length, then even a 2D modeling approach may not be viable due to the length of the required computational time to run such an event on a river system of significant size.

This type of situation is when either 1D modeling or combined 1D/2D modeling may be a better choice. Then for period of record analyses, in which one or more years of simulation are required, generally 1D models are used, but possibly combined 1D/2D models, if the 2D flow areas are either small or they only come into play during large flow events within the period of record (i.e., 2D flow areas are used to model the areas behind leveed systems).

Commonly, 1D models, with large spatial and temporal extents, are generally run on single processors on desktop machines. On the other hand, 2D and 3D models, with spatial and temporal extents larger than tens of miles and a few days, require the utilization of multi-processor machines, such as workstation level personal computers (with many cores) and high performance computing (HPC) systems. Recent developments in 2D and 3D models have allowed for the simulation of hundreds of miles and years of simulations, but these require the utilization of significant computational resources (i.e. super computers).

Required Model Outputs

Almost all studies requiring a hydraulic model need computed water surface elevations, depths, and flow rates. All of the modeling approaches can produce this type of output, but at varying levels of accuracy. Specifically, 1D models compute averaged water surface elevations at each cross section and storage area within the model. Conversely, 2D and 3D models have spatially varying water surfaces based on the size and number of cells/elements/nodes used in the computational mesh. Depths can be computed from any of the model's resulting water surface elevations and inundation maps, but are dependent on the accuracy of the underlying terrain model. Flow rates from 1D models are generally reported as either total flow at each cross section, or the flow rate in the main channel, left overbank, and right overbank (though the flow rate can be further partitioned based on the conveyance across the cross section, and the assumption that the flow is perpendicular to the cross section). Flow rates from 2D models can be acquired along any user defined line within the computational mesh.

Many studies also require velocity information for various reasons. Specifically, 1D models only produce horizontally and vertically averaged velocities. These velocities are often reported separately for the main channel and the left and right overbank areas. Just as with flow, velocities can be further discretized based on cross section conveyance and the assumption that the flow is perpendicular to the cross section. However, in zones of detailed contractions and expansion, for example flow through a bridge opening, velocities produced by 1D models are not as accurate as 2D and 3D models. Detailed velocity distributions for normal channels/floodplains, as well as detailed velocities through contractions/expansions, around sharp bends, and around hydraulic structures requires 2D and possibly even 3D modeling approaches. However, the modeler must develop a computational mesh with enough cells/faces/elements to produce a detailed velocity distribution for the desired locations and structure types.

Other types of information may also be required of hydraulic models. Generally most 1D hydraulic models output a wide range of hydraulic variables at each cross section and hydraulic structure (HEC-RAS outputs close to 300 different hydraulic variables at each cross section for each flow rate/time step). On the other hand, 2D and 3D models generally do not produce this type of output directly, but may have ways to get to the output from post-processing the basic model results of depths, water surface elevations, and velocities.

Other types of output, such as arrival times; flow/depth durations; percent time inundated; residence times; etc., all require unsteady flow modeling, which may be in the form of a 1D, 2D, or 3D unsteady flow model.

Experience of the Modeler

How much experience, and the type of experience, the modeler has will also affect the choice of model being used. In general, 1D steady flow models are the easiest to use and understand the results. Moving into unsteady flow modeling requires more knowledge of hydraulics and also more knowledge of numerical solutions algorithms, such as finite difference, finite volume, and finite element solution techniques. Unsteady flow modeling requires more knowledge of wave propagation, and of how a hydrograph will change in shape as it moves from one point in the system to another. Numerical solution techniques for 1D unsteady flow modeling tend to be either finite difference or finite volume methodologies. Further, 2D and 3D models are mostly either finite element or finite volume approaches.

Understanding these numerical solution techniques is important when using such models. Choosing an appropriate computational time step, cross section spacing (1D modeling), or cell size (2D and 3D modeling), is important to achieving numerical solutions that do not artificially attenuate (often called numerical diffusion) the hydrograph as it moves through the system. Subsequently, 2D and 3D unsteady flow modeling also requires the user to have an understanding of turbulence modeling and its effects on the flow field. Additionally, external forces on the system such as the earth's rotation (Coriolis effect), and wind stresses may be important and can only be accounted for in 2D and 3D modeling approaches.

Chapter 3

Data Requirements

Data requirements can vary significantly for one-dimensional (1D), two-dimensional (2D), and three-dimensional (3D) modeling approaches, as well as steady versus unsteady flow modeling. The amount and quality of available data may dictate the type of modeling that can be accomplished. The main areas in which data requirements can be different are: topographic information (terrain data); channel and floodplain vegetation or land use (defining roughness values); underground drainage infrastructure; surface structure information; required hydrology; boundary conditions; and calibration data.

Terrain Data

Terrain requirements can vary from defining a model with cross sections only (1D modeling) to detailed terrain models of the entire channel and floodplain, as well as features such as: roads, levees, floodwalls, channel training structures, etc. Steady and unsteady 1D models can be driven by only having cross sections at the necessary locations for computing an accurate water surface and routing of the hydrograph. However, 2D or 3D modeling requires a terrain model (Digital Elevation Model, DEM or Digital Terrain Model, DTM) of the entire system.

Additionally, the accuracy of that terrain model will directly impact the accuracy of the 2D or 3D modeling approach. For example, if a detailed terrain model is developed from LIDAR data, but the underwater channel data (bathymetry) is not defined accurately, it is much easier to modify cross sections in a 1D modeling framework, than modifying the entire terrain model to incorporate the channel bathymetry.

In some cases, a necessary task may be to merge bathymetric data from multi-beam Sonar with terrestrial and aerial LIDAR data to produce a seamless terrain model. Specifically, 2D and 3D models will only be accurate if the terrain includes an accurate depiction of the underwater terrain of the main channel, and any structures that influence the flow field. See Figure 3-1 and Figure 3-2 for an example of terrain data with and without channel data burned into the terrain model.

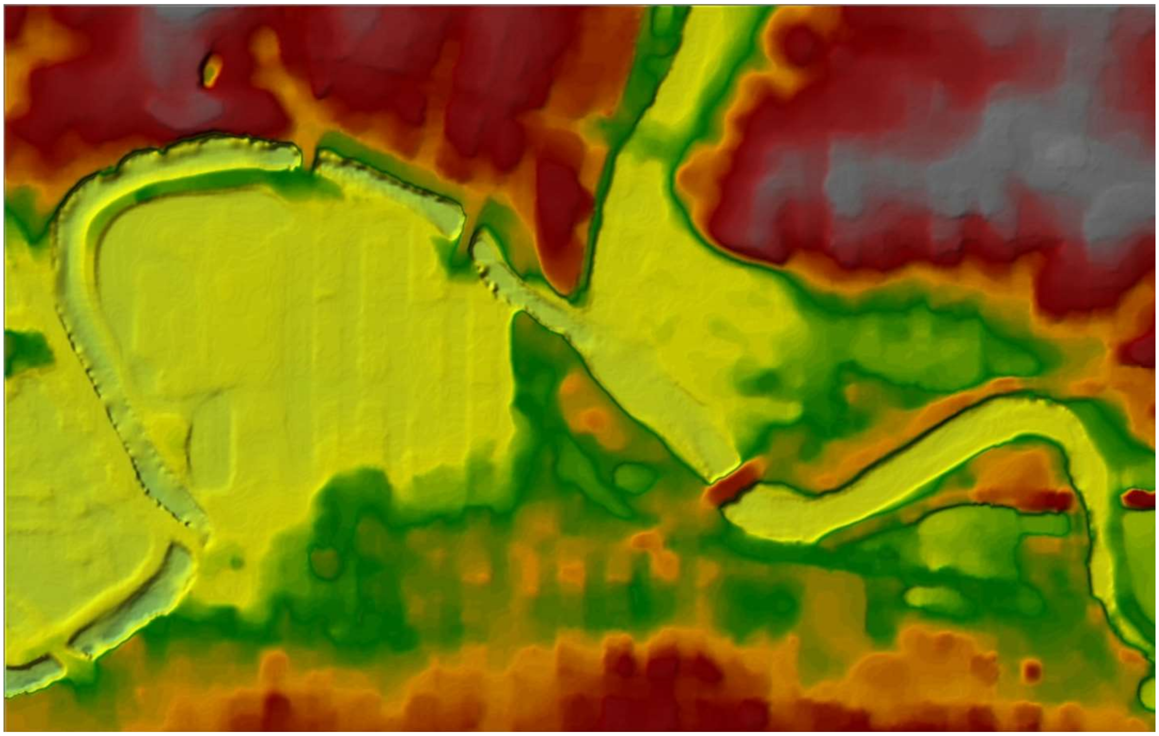

Figure 3-1. Terrain model without under water channel data.

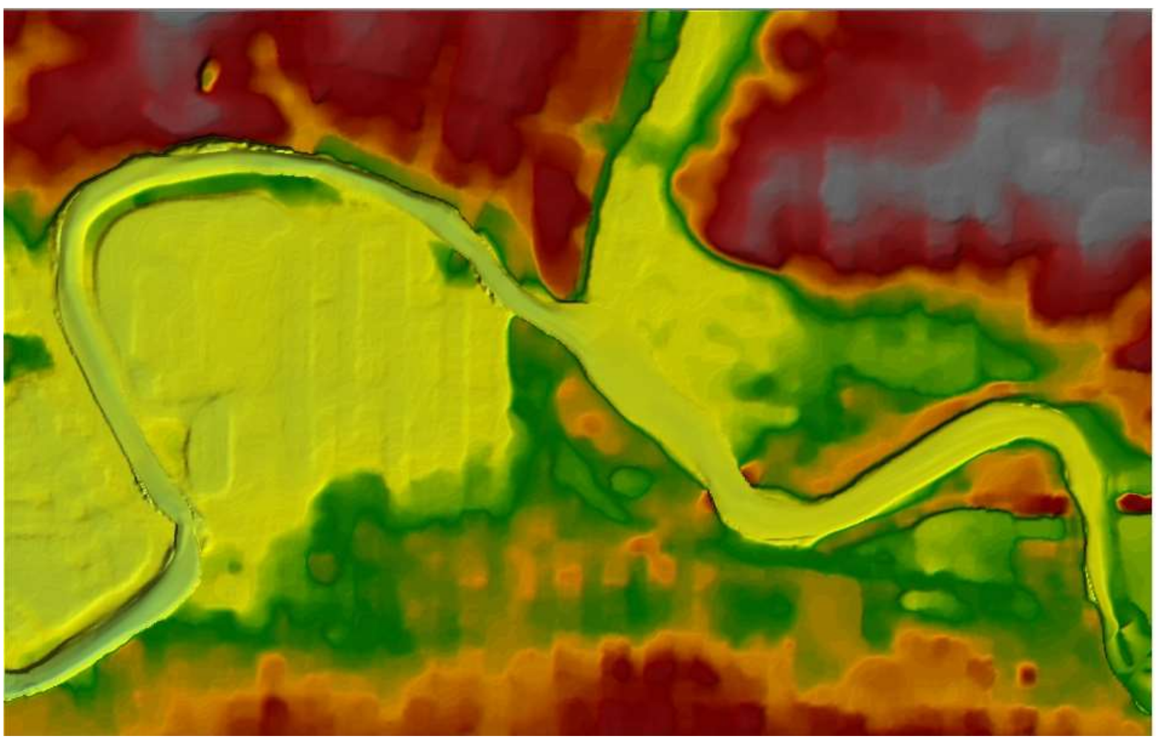

Figure 3-2. Terrain model with channel bathymetry burned into terrain model.

Roughness Coefficients

The data requirements for defining roughness can also vary. In general, knowledge of the vegetation and land use for the entire modeling domain is required for all modeling approaches.

However, roughness for 1D modeling approaches only has to be defined at each cross section. For a 1D modeling approach roughness coefficients can be defined on a cross section by cross section basis, or the modeler can layout spatial vegetation/land use information. The land use grids and polygons are related to roughness values and then the roughness values are extracted at the intersection of the cross sections and the spatial roughness layers.

In general, 2D and 3D modeling approaches require the modeler to layout spatial vegetation/land use information and relate that to roughness values. Then roughness is defined spatially for each computation cell/element face of the 2D/3D computational mesh along the terrain surface boundary. Additionally, the main channel roughness must be defined with separate user-defined polygons. This is generally required because most land use/land cover datasets do not define the channel in detail, and only define it with a single land use type. An example of a 2D model with roughness being defined with land use in the overbank areas, and user defined polygons for the main channel is shown in Figure 3-3.

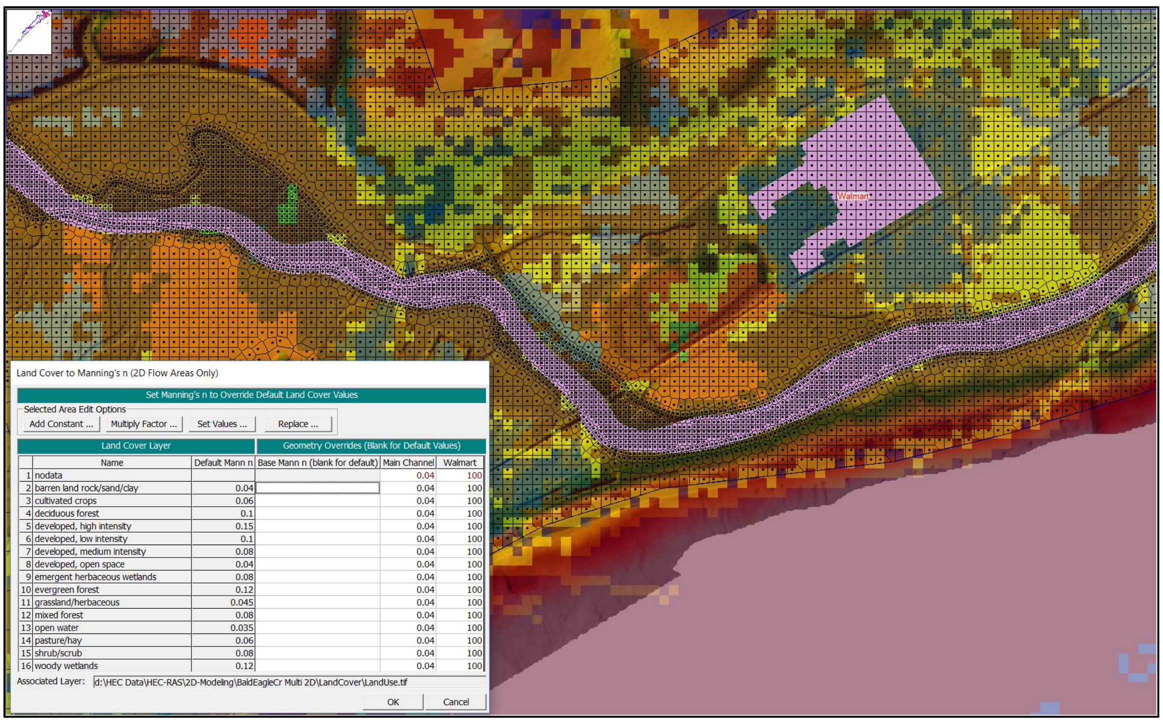

Figure 3-3. Example of land use and user defined polygons to define roughness for a 2D model.

Underground Drainage Systems

Detailed modeling of underground drainage systems, may or may not be necessary for a particular study. For detailed models of urban areas, modeling of the underground drainage systems is often required. Because of the complex interconnections of subsurface pipe systems, steady flow modeling is generally not an option. Most often 1D unsteady flow modeling is used to model subsurface drainage systems. This generalization is due to the fact that the flows and velocities in these types of systems are very one-dimensional in nature. Therefore, 2D and or 3D modeling of existing underground drainage systems is rarely done. Regardless of the modeling approach, the data required to model underground drainage systems is the same.

Hydraulic Structures

The data requirements for modeling hydraulic structures will also vary between 1D, 2D, and 3D modeling approaches. For 1D models, structures can be defined with semi empirical equations or rating curves, and inserted as internal boundary conditions between cross sections. The data required for modeling hydraulic structures is based on the hydraulic computational model being used to model the structure (for example, a weir equation only needs a centerline profile of the top of the weir, the weir shape, and a weir coefficient. While a gate only needs a width, height, invert elevation, and a gate coefficient).

Furthermore, 2D modeling approaches can also use similar hydraulic structure modeling inside of the 2D domain; however, this is then a 1D approach to modeling the structure inside of the 2D area. True 2D/3D flow modeling of a hydraulic structure will require detailed terrain/surface modeling of the hydraulic structure from the upstream entrance, through the structure, to the downstream exit. Additionally, the roughness of the entire structure surface needs to be defined more accurately. Shown in Figure 3-4 is an example of a very detailed 2D model of the 17th Street outfall structure in New Orleans, LA. This model was used as a screening tool to narrow down the number of possible designs to be modeled in more detail with a 3D model.

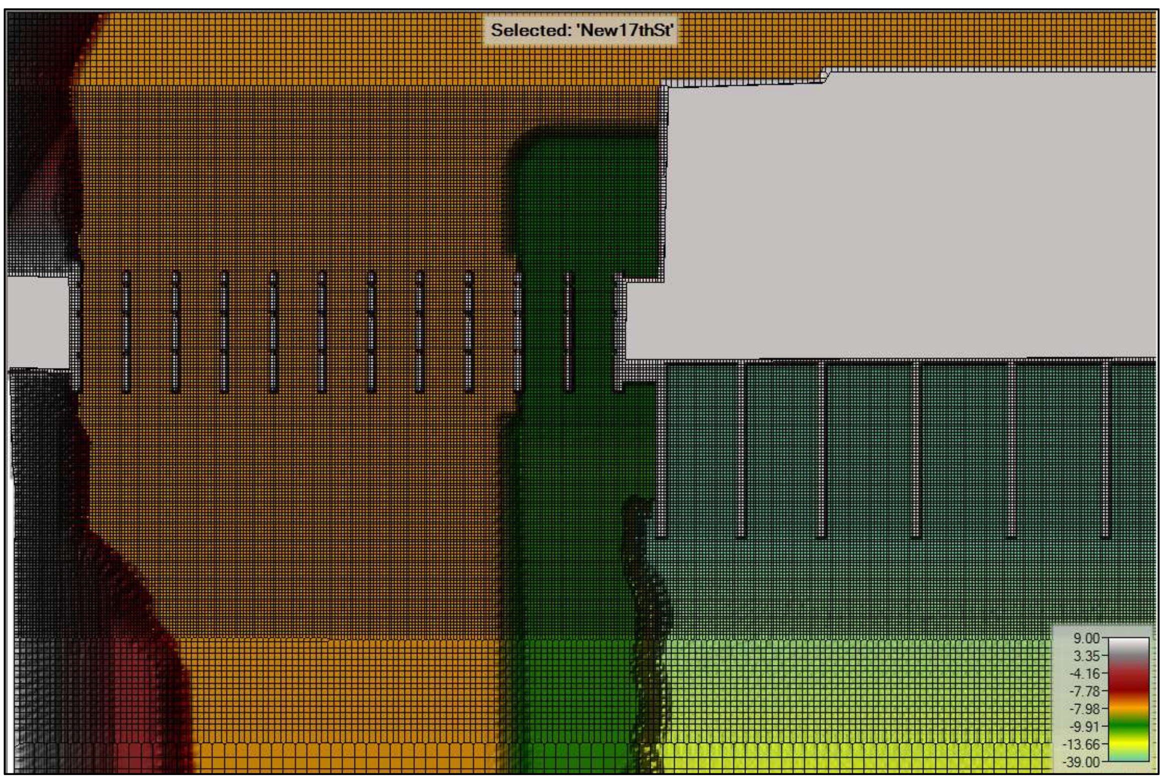

Figure 3-4. Detailed terrain and 2D modeling mesh of the 17th St. outfall canal in New Orleans, LA.

Near-field flows at hydraulic structures may be non-hydrostatic. In such cases, application of hydrostatic models will tend to overestimate energy losses in the vicinity of the structure and may fail to accurately reproduce prototype flow patterns immediately downstream of the structure.

Hydrology

The hydrology used to drive steady flow vs unsteady flow models is different. Steady flow requires the user to define the flow rates for the entire system using either a hydrologic model or measured data. Hydrologic modeling uses simpler routing techniques than hydraulic routing (such as: Muskingum, Modified Puls, and Muskingum-Cunge). Unsteady flow models use hydrographs at all the external upstream boundaries, as well as for any required lateral and internal locations. And unsteady flow models use more detailed hydraulic routing methods to solve how the water moves through the system. The required hydrology for 1D, 2D, and 3D models is virtually the same, as it is much more based on the size/extent of the modeling domain and not the modeling approach. However, how you enter the flow data into the system may be different, depending on the modeling approach (i.e. 1D vs 2D internal boundary conditions).

Boundary Conditions

Downstream and internal boundary conditions can also require different amounts of data for steady versus unsteady flow modeling approaches. Steady flow models generally only need downstream water surface elevations for each profile to be computed. Unsteady flow models may require entire stage hydrographs or stage-discharge rating curves. The amount of data depends on the location and type of boundary condition being applied. If a model is tidally influenced, then full stage hydrographs are required for unsteady flow modeling approaches. However, at river locations that are more controlled by the river flow rate and gravity/frictional forces, rating curves or normal depth (Manning's equation) boundary condition approaches can be applied for steady flow and unsteady flow in the same manner.

Calibration Data

Calibrating the model is required regardless on the model type. The amount and type of observed data required for model calibration can also vary between modeling approaches. Steady flow models use maximum water surface elevations and optionally velocity magnitudes to compare against computed values. Unsteady flow models need entire flow and stage hydrographs at gages, as well as high water marks where available.

Generally, 1D modeling approaches, only make use of observed water surfaces and flow hydrographs at gages, and then high water marks between gaged locations. However, 2D and 3D models may also need observed velocity or flow distribution information. Additionally for 2D modeling, observed velocity needs to be measured spatially across the river and floodplain in order to calibrate the computed velocity distribution. Inundation extents at particular flows are also often required for calibrating more detailed 2D models. Users will need to refer to the documentation that is specific to the model they are using, for further information on calibrating that model.

Chapter 4

Model Output/Results

Requirements for hydraulic model output/results, as well as level of detail, will influence the type of model used for a study. For example, if detailed velocities are needed at the toe of a levee, or around a bridge pier or abutment, then 1D modeling cannot provide that kind of detail, and 2D or 3D modeling will need to be used. So the questions that modelers should ask at the beginning of a study are: what are all of the required hydraulic results needed for this study, what level of detail is needed for each of the hydraulic results, and what level of accuracy is expected/desired for each hydraulic results?

Varying Levels of Detail in Hydraulic Outputs

The following is a table of common hydraulic outputs/results that are often requested from hydraulic modeling studies (Table 4-1). Additionally, Table 4-1 describes level of detail from 1D, 2D, and 3D models for that hydraulic output.

Table 4-1. Hydraulic Model Outputs and 1D, 2D, and 3D Level of Detail.

Hydraulic Output/Results	1D Unsteady Flow Modeling	2D Unsteady Flow Modeling	3D Unsteady Flow Modeling
Max Water Surface Elevation (WSE)	Single average WSE per cross section and storage area.	Horizontally varying WSE. One WSE for each cell.	Horizontally varying WSE. One WSE for each cell/node.
Stage Hydrographs	Average WSE vs. time for cross sections and storage areas.	Horizontally varying WSE vs. time for each computational cell/node.	Horizontally varying WSE vs. time for each computational cell/node.
Peak Flow Rates	Peak flow at each cross section and hydraulic structures.	Peak flows at user defined output/profile lines and hydraulic structures.	Peak flows at user defined output/profile lines and hydraulic structures.
Flow Hydrographs	Flow vs time at each cross section, boundary conditions and hydraulic structures.	Flow vs. time at user defined output/profile lines, boundary conditions, and hydraulic structures.	Flow vs. time at user defined output/profile lines, boundary conditions, and hydraulic structures.
Velocities	Average velocities for main channel, left overbank and right overbank. Further discretization is based on conveyance based subdivisions.	Horizontally varying but vertically averaged velocities. One average velocity for each cell/element face.	Horizontally and vertically varying velocities. One velocity per computational mesh face.

Hydraulic Output/Results	1D Unsteady Flow Modeling	2D Unsteady Flow Modeling	3D Unsteady Flow Modeling
Flow directions and patterns	Flow direction must be defined by the modeler when laying out river reaches and storage areas.	Horizontal flow direction is computed based on the details of the terrain and computational mesh. Horizontal circulation patterns (eddy's) can be ascertained.	Three dimensional directions and flow patterns are computed directly.
Flood Arrival Times	Flood arrival times are based on the computations of 1D average velocities and interpolation of water surfaces between cross sections. Level pool routing cannot be used for estimation of flood arrival times in storage areas.	Flood arrival times are based on two dimensional velocities and flow patterns, as well as water surface elevations within each cell/node.	Flood arrival times are based on three dimensional velocities and flow patterns, as well as water surface elevations within each vertical cells. 3D modeling is currently not used that often for arrival times in riverine situations, due to the heavy computational requirements/times.
Hazard Mapping Depth x Velocity	Depth is computed from spatially interpolated water surface elevations minus the terrain elevation at that location. Velocity is interpolated from interpolating 1D averaged velocities described above.	Depth is computed from cell water surface minus terrain elevations at each location. Velocity is interpolated from 2D spatially computed velocities at each cell/node Face.	Depth is computed from cell/node water surface minus terrain elevations at each location. Velocity is vertically averaged at each location.
Inundation Boundaries	Water surface boundary is computed at each cross section, then an interpolation surface is made and intersected with the terrain to find the water boundary (zero depth elevation).	The zero depth boundary is computed for every cell/node that is partially wet. These boundaries are merged to make continuous polygons.	The zero depth boundary is computed for every cell/node that is partially wet. These boundaries are merged to make continuous polygons.
Shear stress computed as: (γ R S_f).	For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. Values are interpolated between cross sections using the cross section interpolation surface.	For 2D cells/nodes it is the average shear stress across each face, then interpolated between faces.	Hydraulic Properties are vertically averaged, then the average shear stress is computed across each face, then interpolated between faces.
Stream Power computed as: average velocity times average shear stress	For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. Values are interpolated between cross sections using the cross section interpolation surface.	For 2D cells/nodes it is the average velocity times average shear stress across each face, then interpolated between faces.	Hydraulic Properties are vertically averaged, then the average stream power is computed across each face, then interpolated between faces.

Displayed in Figure 4-1 is a plot of the water surface elevation (WSE), at the same location, from a 1D and a 2D model (same flow rate). As shown in Figure 4-1, the 1D model has a horizontal (blue) line across the entire cross section for the water surface. However, the water surface varies from the 2D model (green line). This example location is at the upstream end of a bend to the left, which is why the 2D model results are showing a higher water surface on the right hand side of the terrain profile.

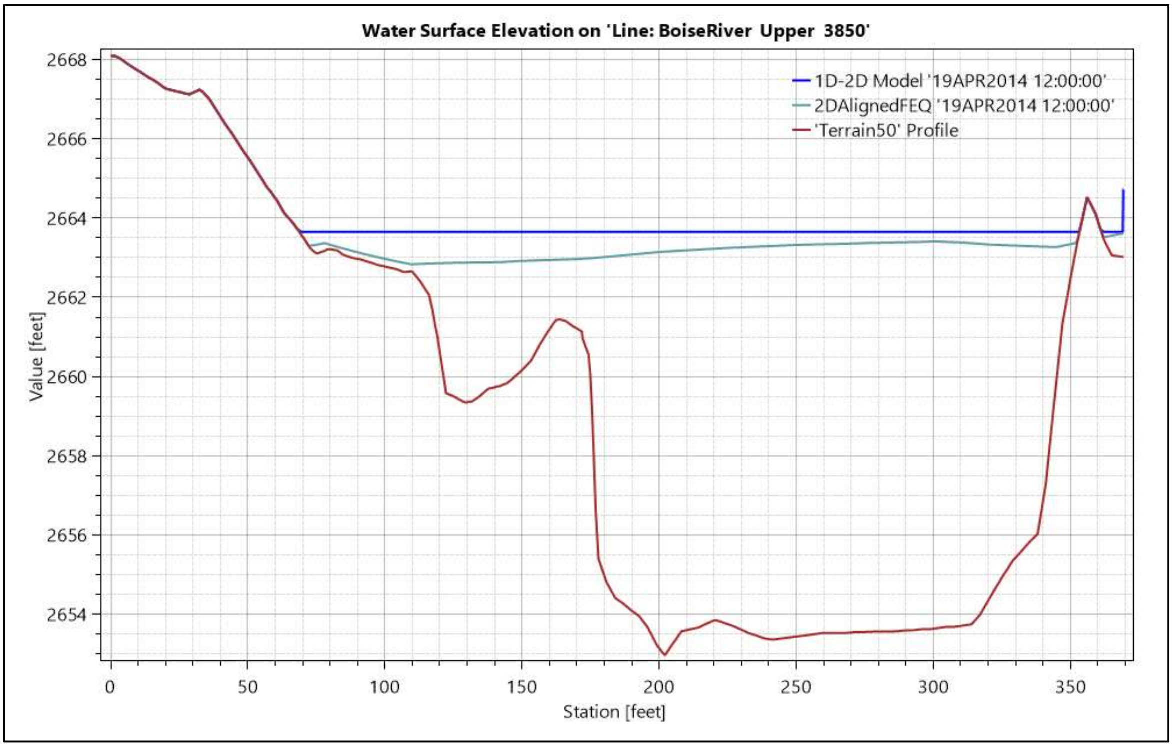

Figure 4-1. Example 1D vs 2D Water Surface Elevation Plot.

Velocity plots from 1D (top plot) and 2D (bottom plot) model results are displayed in Figure 4-2 and Figure 4-3 for the same example location and flow rate. As stated previously, this example location is directly upstream of a bend (to the left) in the river. Figure 4-2 and Figure 4-3 illustrates that the water surface and velocities for the 2D model result contains more of the details at this location, where the 1D result shows a much more uniform distribution of velocity, due to the approach applied to distribute a 1D velocity result in space.

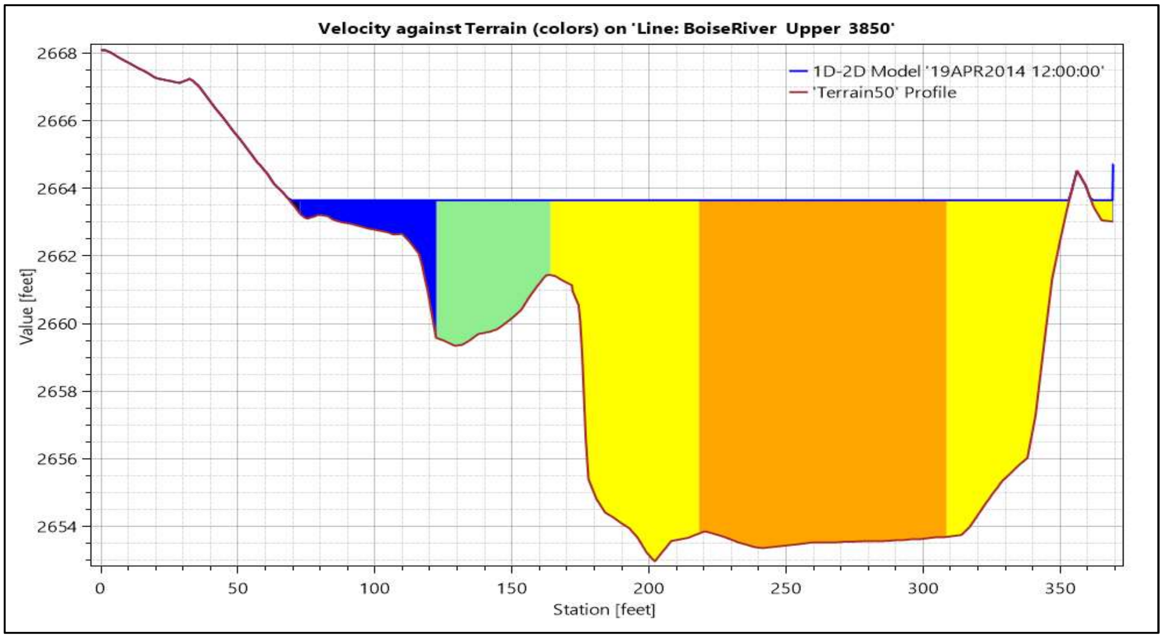

Figure 4-2. One-dimensional (1D) velocity plot at an example cross section.

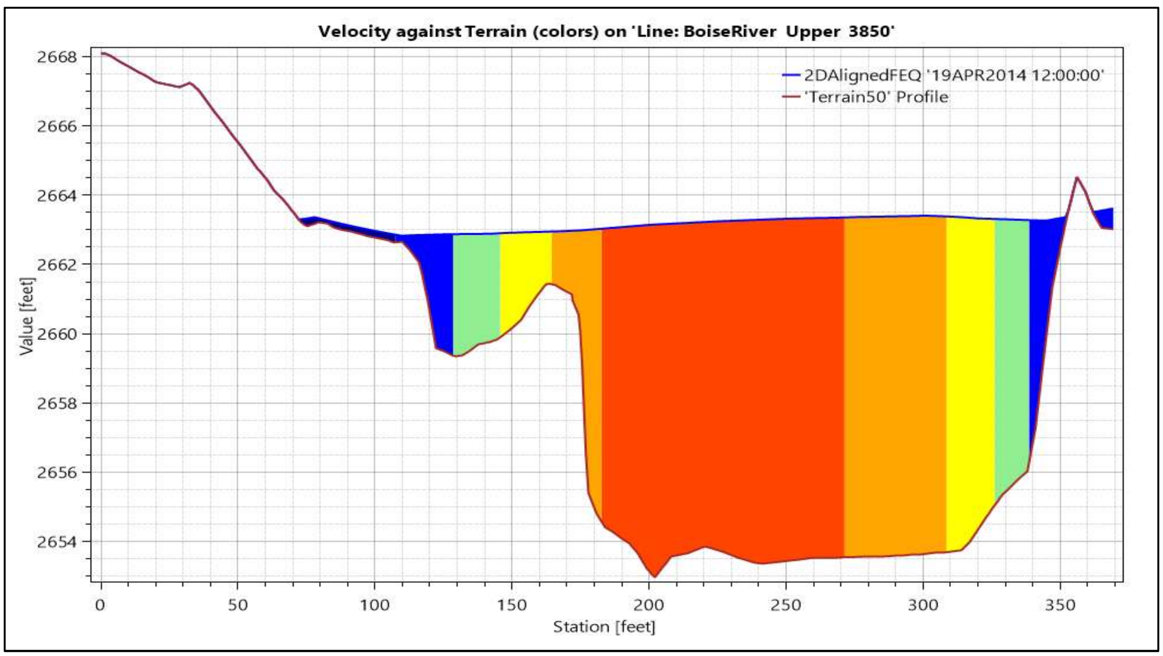

Figure 4-3. Two-dimensional (2D) velocity plot at an example cross section.

Shown in Figure 4-4 and Figure 4-5 are velocity plots from a 2D and 3D model for the same set of gates, at the same flow rate. As you can see from the plots below, the 3D velocity plots are a more accurate depiction of the actual fluid movement through the gates. However, that level of detail may or may not be needed for any particular study.

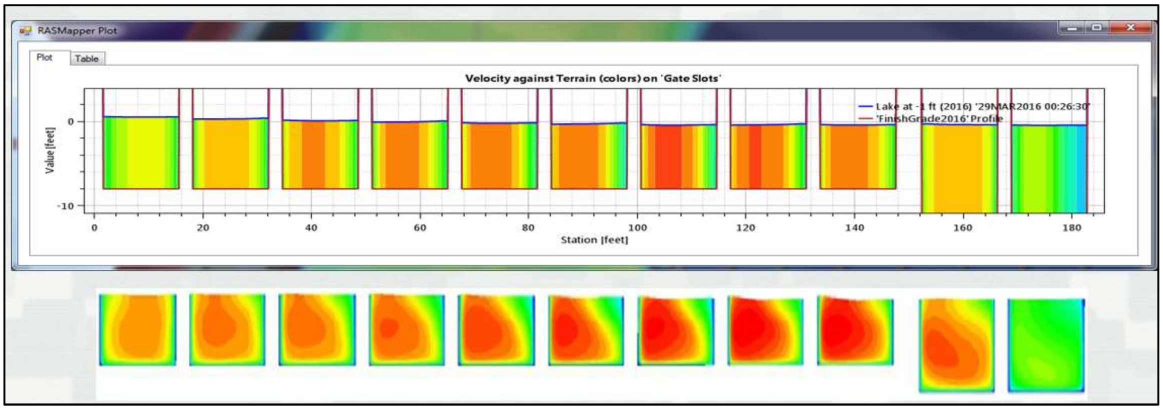

Figure 4-4. Example 2D (top) and 3D (bottom) velocity plots through gate openings.

The results provided in Figure 4-5 and Figure 4-6 are example inundated area maps for an interior area protected by a levee. Specifically, Figure 4-5 displays the results from a 1D model in which the interior area was modeled with interconnected storage areas. On the other hand, Figure 4-6 provides the results from a 2D model of the same area and same flow coming through an upstream breach of the levee.

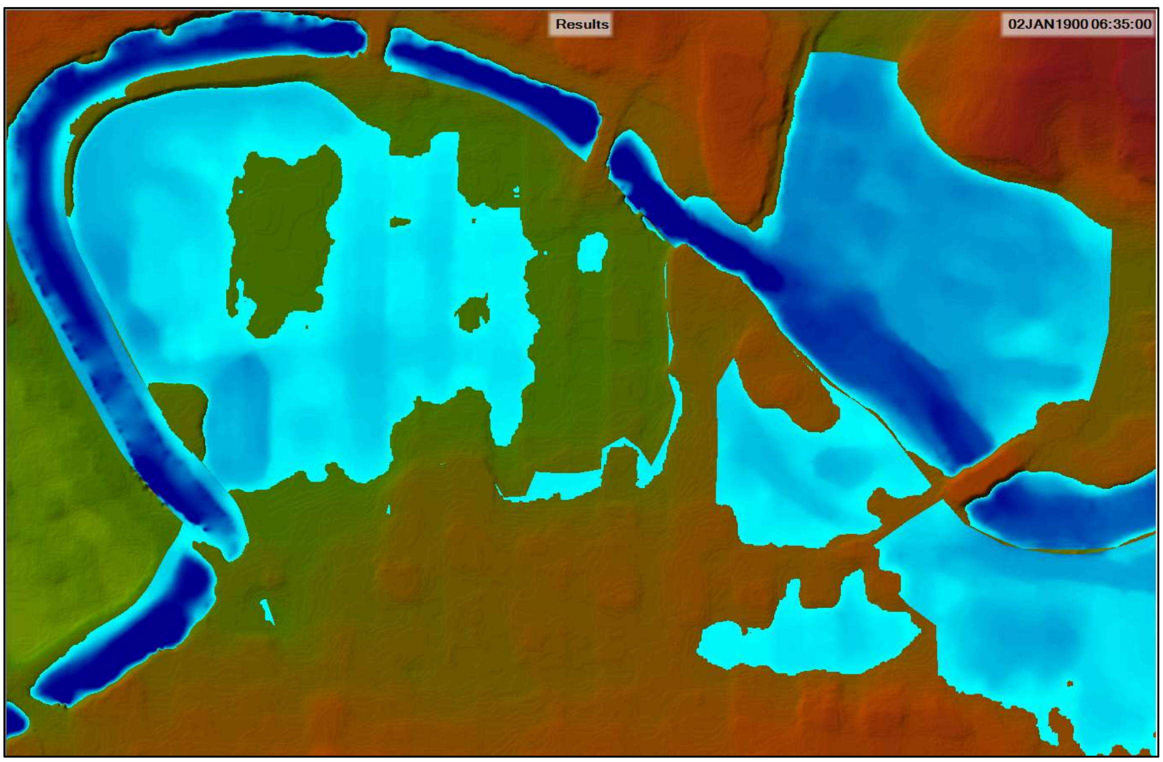

Figure 4-5. One-dimensional (1D) model results for an interior area with a levee breach. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth).

As displayed in Figure 4-5, the 1D results show disconnected water. This is due to the fact that storage areas automatically fill up from the lowest elevation to the highest, with a horizontal water surface. For the same location, results for the 2D model (Figure 4-6) show overland flow paths connecting all of the interior areas. So for this example area, the 2D model provides results that are more realistic for how the flooding would occur, as well as for computing depths, velocities, and arrival times of the flood waters.

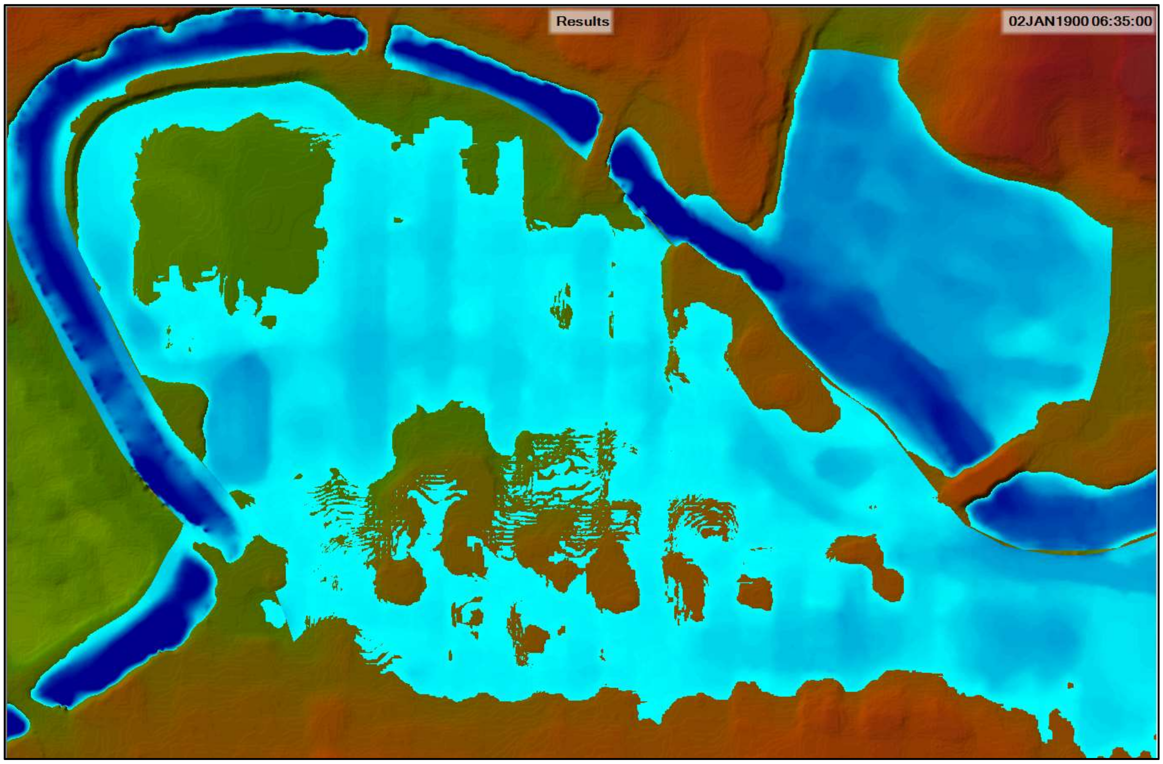

Figure 4-6. Two-dimensional (2D) model results for an interior area with a levee breach. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth).

Expected/Desired Level of Accuracy

Different types of studies have different levels of expected/desired accuracy. For example, in emergency situations, it may be necessary to develop a rough model very quickly with limited data. The expected level of accuracy of such a model is not high, and therefore the modeling approach can be less detailed (i.e., 1D or 2D modeling with less detail). Providing some level of hydraulics results (inundation maps, arrival times, and velocities) is better than having no information at all. Conversely, if the modeler is performing a very detailed design study, then the expected/desired level of accuracy is very high. For this type of study, more time should be spent acquiring detailed data; performing detailed modeling; completing calibration/verification analyses; and completing risk and uncertainty analyses.

The expected/desired level of accuracy will also affect the modeling approach. For example, if the study is expected to have accurate two and three dimensional velocities at a location, then using 2D and 3D modeling approaches will be necessary, as 1D modelling cannot provide high levels of accuracy for that type of hydraulic output. Therefore knowledge of all the required hydraulic outputs, as well as the expected/desired level of accuracy is very important to making a modeling approach decision.

Chapter 5

Steady Flow vs Unsteady Flow Modeling

This chapter discusses the differences between steady and unsteady flow modeling. Specifically, this chapter describes: the definition of steady and unsteady flow; assumptions used in steady flow modeling; hydrologic verses hydraulic routing; differences in the hydraulic calculations; differences in calibration strategies; and steady flow modeling limitations.

Definitions

Steady flow modeling is based on using a specific set of flow rates spatially, then computing water surface elevations, velocities, etc., based on those flow rates. By a strict definition:

Steady Flow – flow (i.e., depth, velocity, discharge) does not change with time.

Because river flows are typically turbulent and the velocity at any point is constantly fluctuating, the definition of steady flow may be expanded to include flows where the mean velocity and depth at any point may be treated as constant over the time period being modeled. Likewise, an assumption of steady flow may be reasonable for hydraulic calculations if flow changes gradually with time. Some examples of steady flow are:

- Flow in a canal at a constant discharge from upstream.
- Natural flow in a river in which the discharge changes gradually with respect to time.
- Modeling a very short reach of river, such that the flow rate is effectively constant throughout the reach at any point in time.
- Modeling a river network in which the flow in each network segment is effectively constant.

Unsteady flow modeling is based on providing full hydrographs at all upstream points in the river system, as well as for any lateral inflow points, then the unsteady flow equations are used to route the hydrographs while simultaneously calculating the water surface elevations. The strict definition of unsteady flow is:

Unsteady Flow – flow (i.e., depth, velocity, discharge) changes with time.

Some examples of unsteady flow are:

- Dam and levee break flood waves
- Tidal Effected bays, estuaries, and streams
- Hydropower releases at reservoirs

- Flash floods
- Tributary flow reversals due to backwater
- Natural floods from rainfall runoff events

Steady Flow Assumptions

Steady flow assumes that a given flow rate persisted for a long enough time, that a steady flow assumption is valid. In general this assumption is true for shorter reaches of a river system, and for events in which the water surface rises and falls slowly. However, as the modeling domain gets larger, and/or flow rates rise and fall quickly, peak flow rates are not occurring at the same time spatially. For these conditions, the assumption of a steady flow rate begins to break down, and may not be appropriate for even computing the water surface elevations.

The slope of the stream is also very important factor in assuming the steady flow assumption. For medium to steep sloping streams, the computed water surface is based on the terrain, roughness, and flow rates in the immediate vicinity of where the water surface is being computed (except when there is significant backwater from a downstream structure or constriction). Therefore, computing a water surface elevation based on maximum flow rates at all locations is a valid assumption even for very large systems, in which the peak flow did not occur simultaneously.

Shown in Figure 5-1 are water surface profiles for a moderately steep stream (5 ft/mile), computed with a steady flow model. Both profiles have a flow of 9,000 cubic feet per second (cfs) upstream. One of the plotted profiles was computed with the peak flows entered at all locations (Figure 5-1). The second profile is based on the instantaneous flow rates at the time the peak flow (9,000 cfs) was at the upstream end of the system. Notice that because the slope is relatively steep, the resulting water surface at the upstream end (where the flow is 9,000 cfs for both runs) is the same (Figure 5-1).

However, as the stream slope flattens, downstream water surface elevations/flow rates will impact the computation of the water surface elevations upstream. The flatter the slope, the greater the distance upstream that will be impacted by downstream water surfaces. For these types of situations, the assumption of steady flow would produce water surface elevations that are too high. The elevation overestimation is due to the fact that a computed water surface upstream, based on a peak flow rate, will be biased by the downstream water surfaces, which is also computed based on peak flow rates. However, if those peak flow rates did not occur at the same time, the steady flow assumption is invalid, and will lead to overestimation of the water surface elevations.

Shown in Figure 5-2 is a water surface profile plot for a flat stream (0.5 ft/mile). Both profiles have a flow of 9,000 cfs upstream (Figure 5-2). One of the plotted profiles was computed with the peak flows entered at all locations (Figure 5-2). The second profile is based on the instantaneous flow rates at the time the peak flow (9,000 cfs) was at the upstream end of the system (Figure 5-2). Notice that because the slope is very flat, the resulting water surface at the upstream end is not the same, even though 9,000 cfs is being used for both profiles at the upstream end. There is over 0.5 ft of difference in the resulting water surface at the upstream end, with the steady flow model of simultaneous peak flows everywhere giving higher answers than the instantaneous flow model (Figure 5-2).

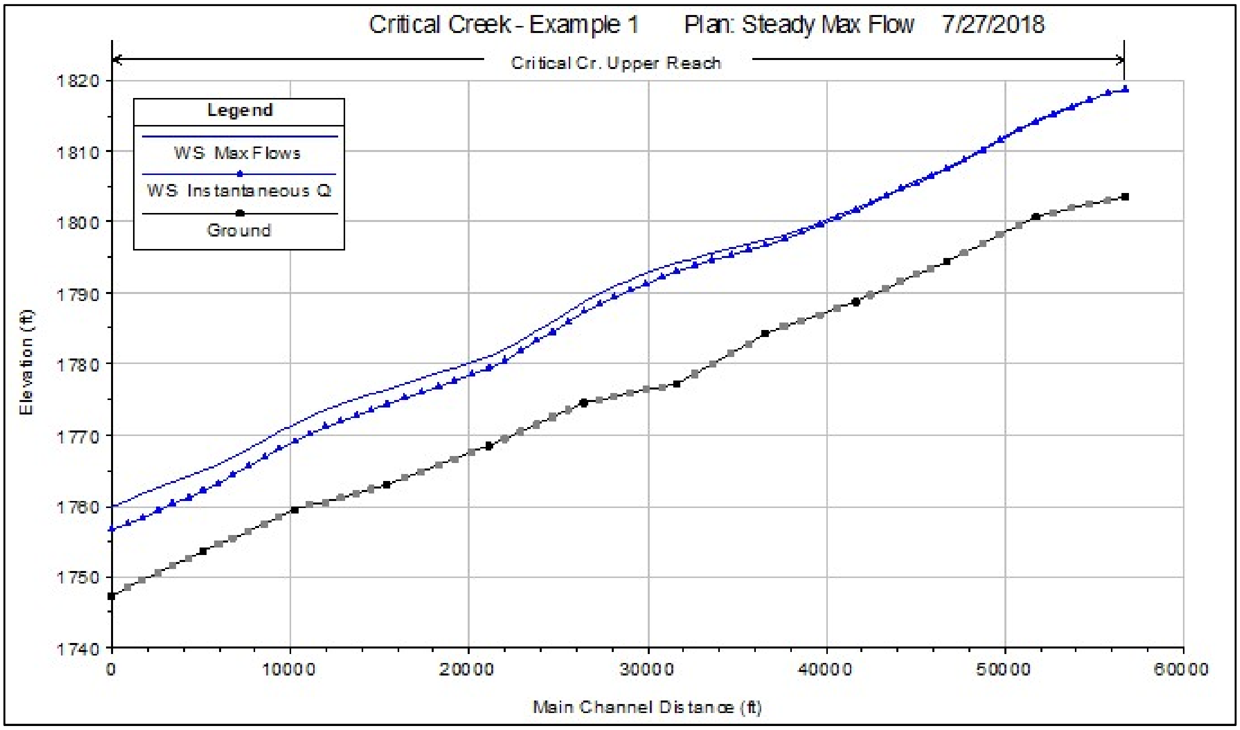

Figure 5-1. Steep stream (slope = 5 ft/mile) with profiles computed using maximum flows and instantaneous flows.

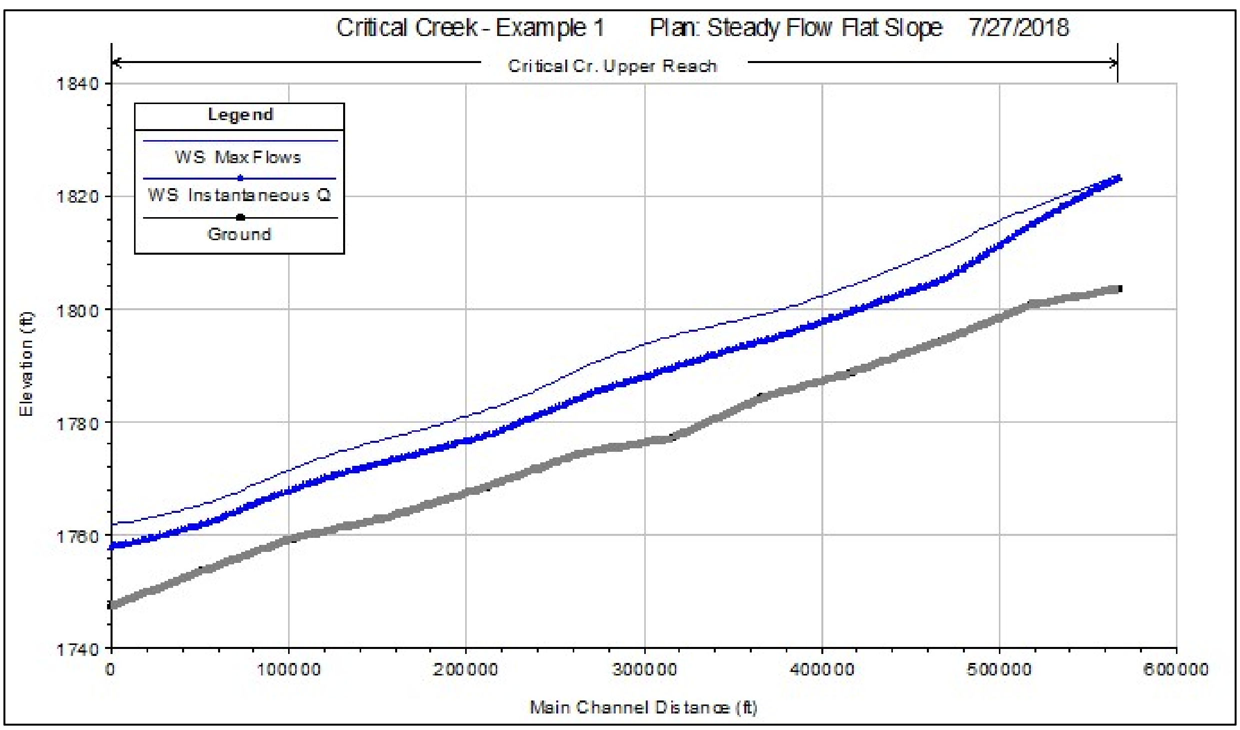

Figure 5-2. Flat stream (slope = 0.5 ft/mile) with profiles computed using maximum flows and instantaneous flows.

Hydrologic vs Hydraulic Routing

A successful application of any steady flow model requires that flow rates have already been accurately computed by a hydrologic model or measured by an accurate and complete set of stream gages (or some other appropriate method). Hydrologic routing consists of solving the continuity equation and a relationship between storage in the river and discharge at the outlet of the routing reach. Some examples of hydrologic routing methods are Modified Puls, Muskingum, and Muskingum-Cunge.

If a hydrologic model is being used to not only compute the precipitation-runoff over the watershed, but perform all of the routing within the system, then the flow rates used in the steady flow model are only as accurate as the hydrologic model. So, the use of a steady flow hydraulic model, is predicated on the fact that a hydrologic model was considered to be appropriate for not only developing the flow rate from precipitation-runoff computations, but also routing all of the flows through the system during the event. Therefore, a large part of the decision of steady flow versus unsteady flow hydraulic modeling comes down to the question: is hydrologic stream flow routing accurate enough to produce flow rates that can be used in the corresponding steady flow hydraulics models?

Hydraulic routing (unsteady flow routing) solves the continuity and momentum equations together, in order to route the hydrographs and compute the water surface elevations.

Computational Differences

In order to better understand the differences between steady flow modeling and unsteady flow modeling, the modeler should be aware of all of the computational differences between the two approaches. The following is a description of the major computational differences between 1D steady and 1D unsteady flow routing.

The unsteady flow equations (hydraulic routing) are more physically based in that they are derived from the continuity equation and Newton's second law of motion:

$$\sum F = ma$$

where:

F = Sum of all the forces acting on a body of water
m = Mass of the body of water
a = Acceleration (or deceleration) of the fluid

Steady and Unsteady Flow Equations

As mentioned previously, the unsteady flow equations are derived from Newton's second law of motion. Shown in Figure 5-3 is a diagram of the forces acting on a body of water in one dimension (i.e., one-dimensional flow).

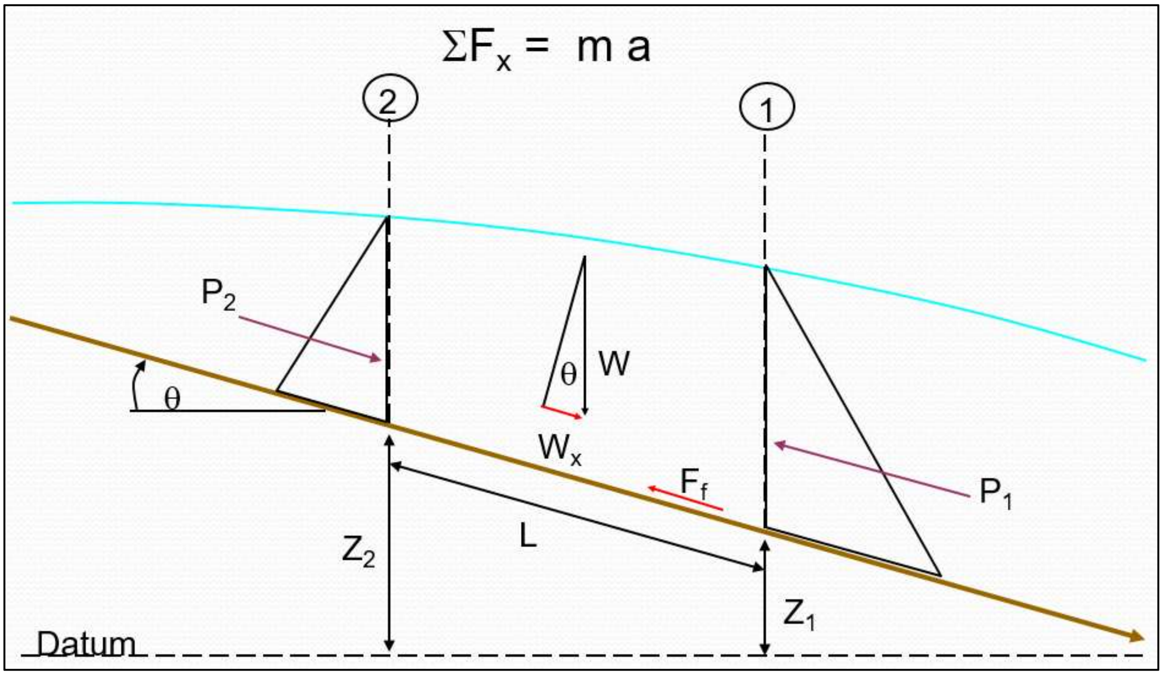

Figure 5-3. Forces acting on a body of water from cross section 2 to cross section 1.

Applying Newton's second law of motion to a body of water enclosed by two cross sections at locations 1 and 2, the expression for the change in momentum over a unit time can be written as:

$$P_2 - P_1 + W_x - F_f = Q\rho\Delta V_x$$

where:

P = Force due to hydrostatic pressure
W_x = Force due to weight of water in X direction
F_f = Force due to external boundary friction from 2 to 1
Q = Discharge
ρ = Density of water
ΔV_x = Change in velocity from 2 to 1 in X direction

The one-dimensional momentum equation and the continuity equation can be written in partial differential equation form, with respect to Discharge (Q), Area (A), and Depth (h), and are commonly shown in hydraulic text books as follows:

Momentum Equation:

$$\frac{\partial Q}{\partial t} + \frac{\partial(\beta\, {Q^2}/{A})}{\partial x} + gA\left(\frac{\partial h}{\partial x} - S_0 + S_f\right) = 0$$

Continuity Equation:

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} = q_l$$

where:

Q	= Discharge
β	= Velocity distribution coefficient
A	= Cross sectional area
t	= Time
x	= Distance in the direction of flow
h	= Depth of water
S_0	= Bed slope
S_f	= Friction slope, from Manning's equation
q_l	= Lateral inflows

For steady flow, the time based terms in the momentum and continuity equations go to zero. Therefore the steady flow form of the one-dimensional momentum and continuity equations can be written as follows:

Steady Flow form of the Momentum Equation:

$$\frac{\partial(\beta Q^2/A)}{\partial x} + gA\left(\frac{\partial h}{\partial x} - S_0 + S_f\right) = 0$$

Steady Flow form of the Continuity Equation:

$$Q = VA$$

While the above form of the continuity and momentum equations can be used to solve for one-dimensional steady flow, in general most 1D steady flow programs solve the one-dimensional energy equation instead. The steady flow one-dimensional energy equation (often called Bernoulli's Equation) is written as:

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

where:

Z	= Elevation of the main channel inverts at cross sections 1 and 2
Y	= Depth of water at cross sections 1 and 2
V	= Average velocity of water (Q/A)
α	= Velocity weighting coefficients
g	= Gravitational acceleration
h_e	= Energy losses from cross section 2 to 1. (friction losses (h_f) and contraction/expansion losses (h_{ce}))
h_f	= Friction losses $h_f = LS_f$
L	= weighted average distance between cross sections
h_{ce}	= Contraction and expansion losses $h_{ce} = C\left\lvert\frac{\alpha_1 V_1^2}{2g} - \frac{\alpha_2 V_2^2}{2g}\right\rvert$
C	= Contraction or expansion loss coefficient.

For one-dimensional steady flow, the energy equation is solved iteratively from one cross section to the next. For unsteady flow, the continuity and momentum equations are solved simultaneously, generally in a matrix solution scheme that solves for all space (all computational points) each time step (implicit solution scheme). However, there are other solution schemes that solve for one cross section at a time (explicit solution schemes).

Additionally, because there are time based derivatives in the unsteady flow equations, the modeler must select an appropriate computational time step to solve the equations. The computational time step is selected based on the resolution of the hydrographs to be routed, as well as numerical accuracy and stability of solving the non-linear mathematical equations. A common approach to selecting the computation interval is to use a numerical accuracy/stability criteria called the Courant condition:

$$C = \frac{V_w \Delta T}{\Delta X} \leq 1.0$$

Therefore:

$$\Delta T \leq \frac{\Delta X}{V_w}$$

where:

C = Courant Number
V_w = Flood wave velocity (wave celerity) (ft/s)
ΔT = Computational time step (s)
ΔX = Average distance between cross sections (1D) or computation cell/element size (2D and 3D)

Hydraulic Properties

Specifically for 1D modeling, when solving the 1D energy equation, all of the hydraulic properties (area; wetted perimeter, conveyance; storage; etc.) are solved exactly for each cross section as needed. Because unsteady flow simulations are much more computationally intensive (the equations are often solved iteratively for thousands of time steps), the hydraulic properties are often pre-computed for all possible water surface elevations at each cross section or bridge/culverts. Hydraulic properties are then interpolated from the pre-computed curves during the unsteady flow computations. Generally, linear interpolation methods are used, so there is some error depending on the number of curves and the number of points in each curve. An example of a family of flow vs. head water elevation curves that is precomputed for a typical bridge crossing is shown in Figure 5-4.

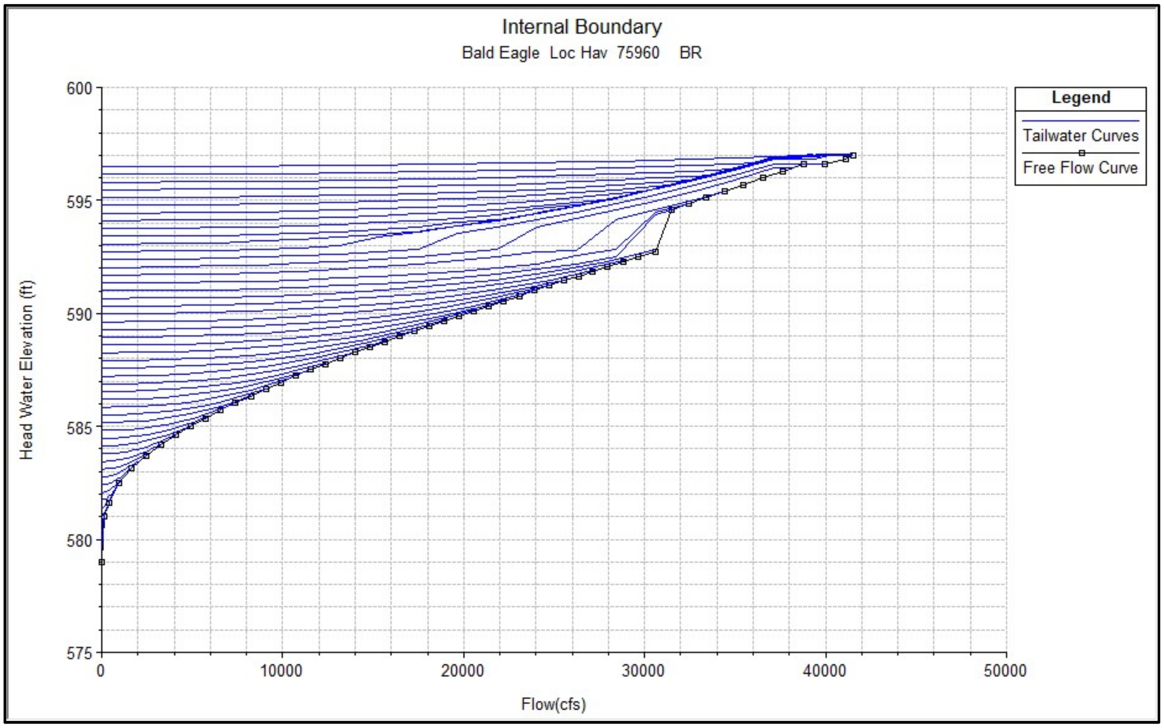

Figure 5-4. Example family of rating curves pre-computed for a bridge.

Friction Losses

For the 1D energy equation, the term h_f measures the internal energy dissipated in the whole mass of water between the two cross sections, whereas the item h_f in the momentum equation measures the losses due to external forces exerted on the water by the walls of the channel. The inherent distinction between the two principles lies in the fact that energy is a scalar quantity whereas momentum is a vector quantity. Ignoring the small differences in the velocity weighting coefficients α and β, for gradually varied flow, the internal energy losses, computed from the energy equation, are practically identical to the external forces in the momentum equation (Chow, V.T., Open Channel Hydraulics, McGraw-Hill, 1959, p. 51.).

Both steady flow and unsteady flow use friction loss equations, such as Manning's equation to describe the internal energy losses and external forces due to friction. In both methods, Manning's equation is used to compute the friction slope term S_f at a point (i.e. cross section or 2D cell face). Additionally, both steady flow and unsteady flow require that an average friction slope be computed between the two cross sections, in order to accurately compute the friction loss over the length between cross sections. As there are different ways of computing an average friction slope, this can be a source of differences between the two computational approaches. For example, HEC-RAS uses a method called the "Average Conveyance" equation to compute the average friction slope for 1D steady flow. However, the average friction slope for unsteady flow is computed with the "Average Friction Slope" equation in HEC-RAS (HEC-RAS Hydraulic Reference Manual, Chapter 2, September, 2016).

Contraction and Expansion Losses

The momentum approach integrates forces acting over the surfaces and ends of a control volume; therefore, impacts of flow contractions/expansions are captured in the forces on the upstream and downstream ends of that control volume. Of course, proper selection of the flow areas through a contraction and expansion (cross section placement) are needed for this approach to work out correctly. On the other hand, the energy approach integrates work/energy for the control volume; empirical coefficients multiplied by the change in velocity head are used to describe the losses associated with the turbulent energy expenditure associated with flow contraction/expansion. However, research has found that using a model calibrated for steady flow within the HEC-RAS unsteady flow solver can result in lower computed water surfaces due to missing the complete losses from contraction/expansion turbulence. Because this is a known computational difference in HEC-RAS between steady flow and unsteady flow, empirical contraction and expansion losses are generally added to unsteady flow computational algorithms as an option. To illustrate this option, Figure 5-5 displays the model results for a calibrated steady flow HEC-RAS model; unsteady flow model results with the exact same geometry and flow data; and then a final calibrated unsteady flow model. The unsteady flow model was calibrated by turning on the empirical contraction/expansion forces, and adjusting the coefficients to match the already calibrated steady flow model for the same flow rates.

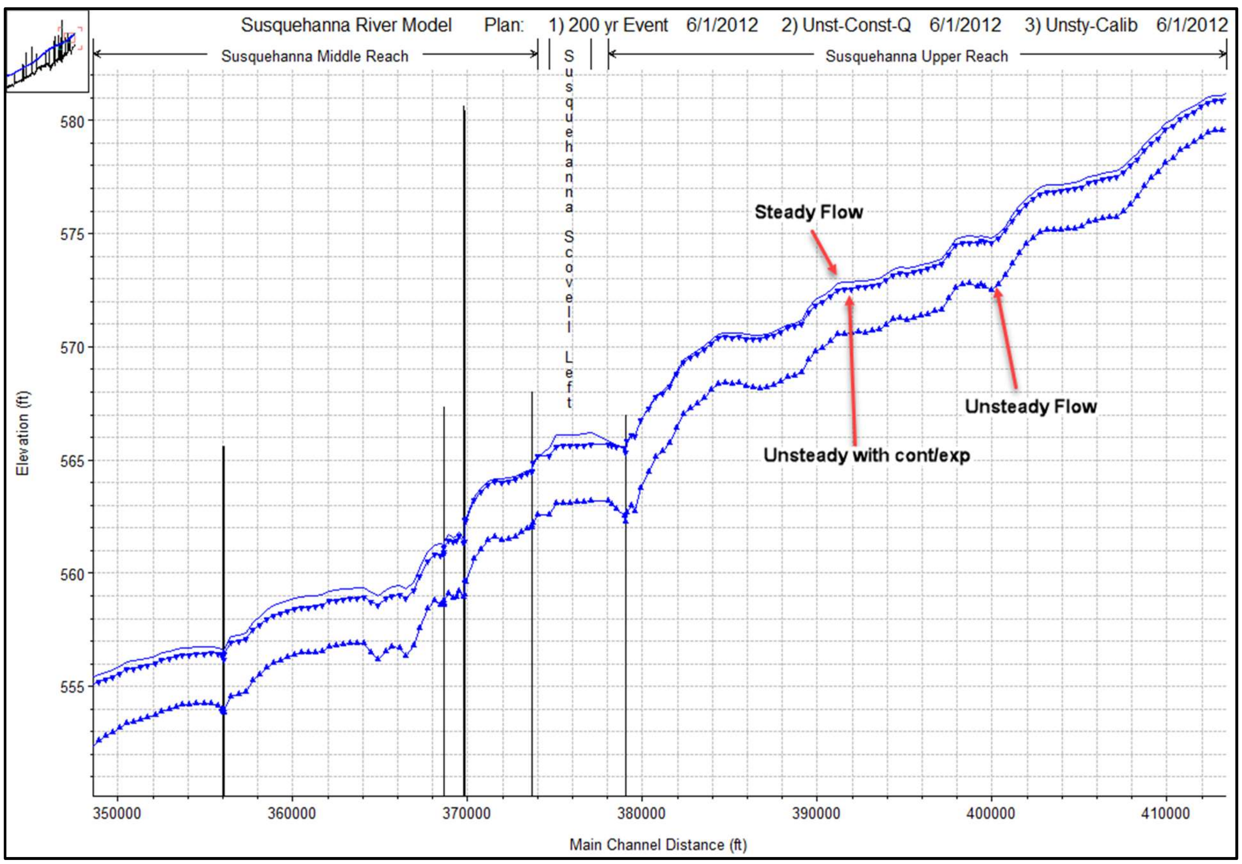

Figure 5-5. Calibrated Steady Flow Model and Unsteady flow model with and without contraction and expansion losses added.

Storage/Ineffective Flow Areas

For one-dimensional (1D) flow modeling, defining portions of the cross sections as ineffective flow areas is required in order to get the correct amount of active (effective) flow area. If this requirement is not done, the flow area will be wrong in many locations, producing too low or too high water surface elevations and velocities, which in turn will affect the computation of friction losses and contraction/expansion losses. For steady flow modeling, the ineffective flow areas are truly only used to describe which portions of the cross section has moving water and which portions do not. Volume accounting is not done in steady flow, so the effects that floodplain storage has on the hydrograph are done outside of the hydraulics model (hydrologic routing). Shown in Figure 5-6 is an example of laying out ineffective flow areas for both 1D steady flow and 1D unsteady flow modeling.

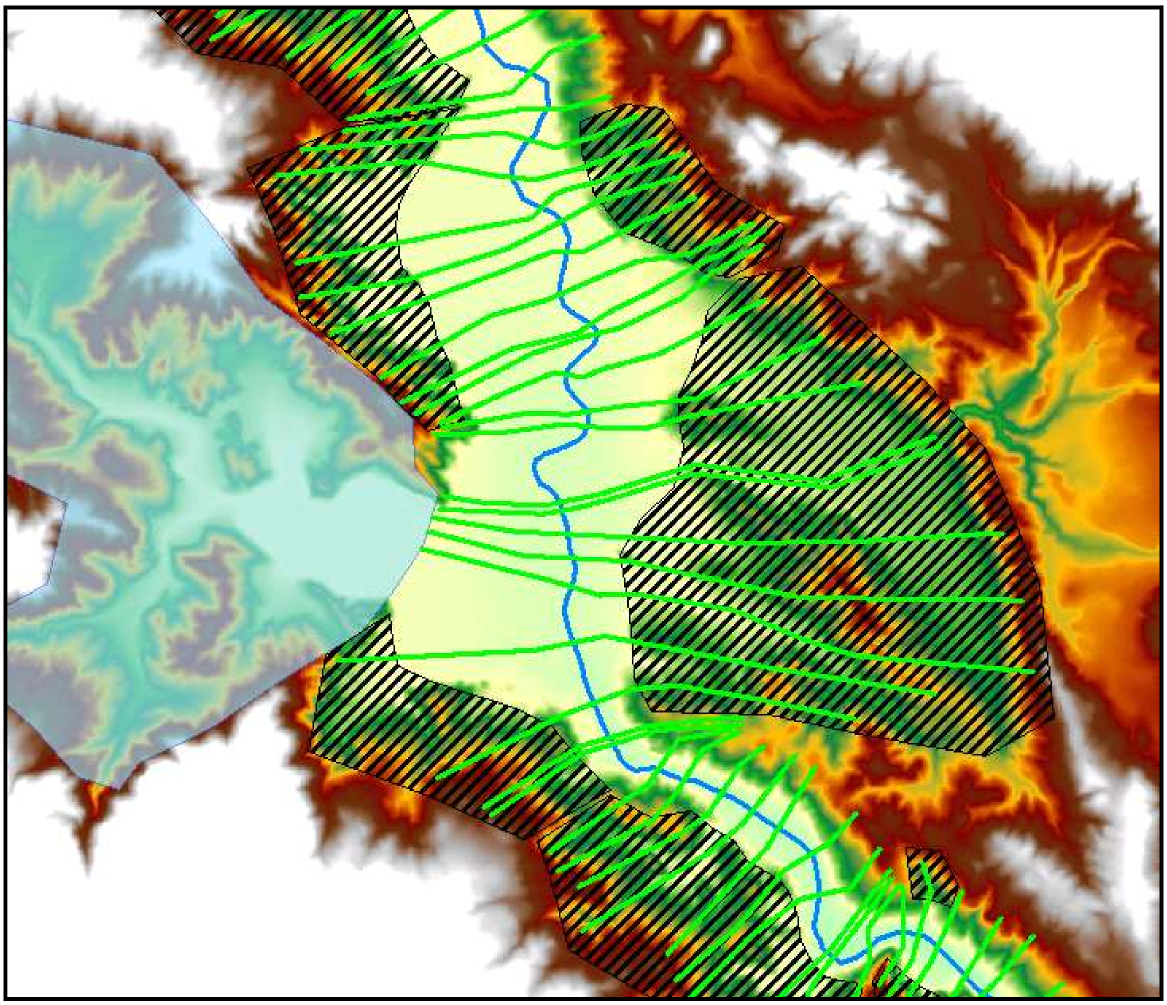

Figure 5-6. Example layout of ineffective flow areas (black diagonal-line-filled polygons) for 1D modeling.

For 1D unsteady flow modeling, ineffective flow areas are not only used to define the ineffective flow area, but they also are used to compute storage volumes between the cross sections. As a hydrograph is routed through a reach containing ineffective flow areas (and therefore storage volume areas), water will go out of the channel to fill up these storage volumes. As the flood wave passes, water will come back out of the storage areas into the channel as flow on the falling limb of the hydrograph. An example of the effect of cross section storage due to ineffective flow areas is shown in Figure 5-7.

In Figure 5-7 there are two hydrographs, one at the upstream end of the reach and one at the downstream end. This example river reach contains a significant amount of ineffective flow areas. As the hydrograph passes through the reach, water goes into storage area. Water on the rising side of the hydrograph and the peak flows into the storage area. Then, as the peak passes, water comes back out of storage and adds flow to the falling limb of the hydrograph.

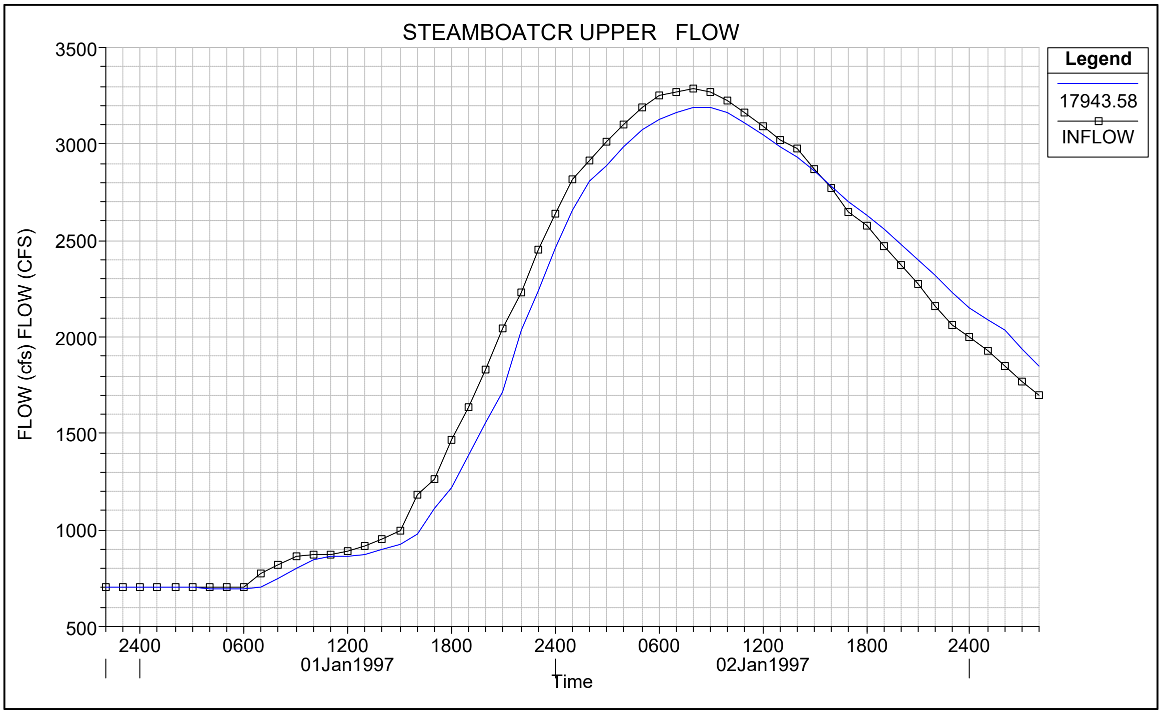

Figure 5-7. Hydrograph going into and out of a river reach with ineffective flow areas acting as storage.

Calibration Strategies

Model calibration strategies vary between steady flow modeling and unsteady flow modeling. In general, different approaches are required due to the fact that there is less computed information to compare with observed data when performing a steady flow modeling approach.

In general, the calibration strategy for 1D steady flow modeling is to compare computed water surface elevations to gaged data and any available high water marks. Adjustments are generally made to Manning's n values, contraction and expansion coefficients, and any hydraulic structure coefficients (weir coefficients, bridge and culvert coefficients, etc.). Additional adjustments may be needed for the location of ineffective flow area stations/elevations and cross section levee elevations. For 2D and 3D modeling, additional calibration of velocities profiles would be performed if observed velocity measurements are available.

For 1D unsteady flow modeling, the calibration strategy starts the same as for steady flow modeling, however, the modeler also compares entire hydrographs (water surface elevation and flow vs time) against computed results. So in addition to looking at peak stages, the entire shape, timing, volume, and magnitude of the hydrographs are evaluated and compared to the observed values. The calibration process is more complex with unsteady flow modeling. Differences in timing and magnitude could be caused by incorrect roughness coefficients, poor definitions of floodplain storage, or even wrong hydraulic structure coefficients. Adjustments made to the model upstream will affect timing and magnitude of the flow/stage downstream. Adjustments made downstream can affect upstream water surface elevations. Additionally, keeping track of volume within the system is very important. Sometimes, the flow boundary conditions provided to the model may not accurately describe the volume coming into the system or the timing of flood peaks.

Steady Flow Modeling Limitations

Steady flow models, or even running an unsteady flow model (1D, 2D or 3D) in a steady flow mode (constant flow), should generally not be used when the following situations exist in the river system being analyzed (this is not an exhaustive list):

- The river is tidally influenced, and the tide has a significant effect on the water surface elevations for the area of interest.

- The events being modeled are very dynamic with respect to time (i.e., dam break flood waves; flash floods; river systems in which the peak flow comes up very quickly, stays high for a very short time, and then recedes quickly).

- Complex flow networks and/or flow reversals occur during the event.

- Dynamic events such as levee overtopping and breaching occur during the event.

- Extremely flat river systems, where gravity, hydrostatic pressure, and friction are not necessarily the only significant force acting on the flow (i.e. local and convective acceleration forces).

- Systems with Pump stations that move a significant amount of water.

- Systems with structures that have complex gate operations based on stages and flows in the system.

- Systems with a tremendous amount of storage in which the hydrograph will attenuate significantly.

Even considering all of what is stated above. There are still many areas in which a good hydrologic model (one that is representative of the watershed and has been well calibrated) can be used in conjunction with a steady flow hydraulics model to perform watershed studies requiring hydraulic model results. However, it is up to the modeler to decide when using a steady flow modeling approach is not appropriate.

Unsteady Flow Modeling Limitations

Initial conditions.

Unsteady flow models generally require the specification of initial estimates of water surface elevation and velocity (i.e., initial conditions) throughout the model domain. These estimates may be obtained in a variety of ways. Some models will compute the initial conditions from a steady flow analysis based on the initial flows and stages specified as boundary conditions in the simulation. The modeler may specify zero inflows and a constant water surface and then drain the model by reducing stages downstream while increasing inflows, an unsteady flow computation, until the desired initial conditions are computed (i.e., a “cold-start”). Another alternative is to obtain the initial conditions from a previous model simulation (i.e., a “hot-start”) or from another model (i.e., “warm-start”).

In most cases, the initial conditions will be an imperfect representation of the system state at the start of an unsteady flow simulation. These imperfections can range from localized, short-term inconsistencies in computed velocity and depth to significant errors in the volume of water present in the model at the beginning of the simulation. Thus, unsteady flow models generally require a “warm-up” or “spin-up” period at the beginning of the simulation to “flush” these imperfections from the system. In the case of a hot-start that simply extends the simulation of a given set of boundary conditions, (i.e., the flow and stage hydrographs specified at the model boundaries) the spin-up period may be negligible. For large, complex model domains or relatively flat rivers, flushing significant volume errors from the system may require a warm-up period on the order of time required to route a flood hydrograph through the system.

It should be noted that water quality transport models generally require additional sets of initial condition data. Transport models may be more sensitive to imperfections in the initial conditions than hydrodynamic models and require longer spin-up periods to compute acceptable starting conditions for a simulation.

Event selection.

As previously described, steady flow models tend to generate conservative estimates of water surface profiles as compared to unsteady flow models. Likewise, a conservative estimate of maximum channel velocities can be generated with a steady flow model by reducing hydraulic roughness coefficients in the channel within reasonable limits. In unsteady flow modeling, the selection of the simulation hydrographs (boundary conditions) used to develop these estimates can have a significant influence on computed peak water surface or velocity profiles.

Consider the following example. In an unsteady flow simulation of flood passage through a typical river, the computed stages for a given discharge will be higher on falling limb of the hydrograph than on the rising limb of the hydrograph (there are exceptions to this typical case).

A simple conceptual model of this “looped rating curve” phenomena is that river stages downstream of an observation location will be lower on the rising limb than on the falling limb.

Now, consider the case where the flood hydrograph is composed of multiple, closely spaced events with the same peak discharge. If the computed downstream stage never recovers to its original condition, we can reasonably expect that each successive discharge peak at our observation point will produce a higher peak stage (and a lower peak velocity). Thus, an estimate of peak stage based on an unsteady flow simulation of a simple flood hydrograph with a single peak could understate actual flood risk in the prototype. In alluvial rivers the observed “loop rating curve” may be larger (wider loop) than the “hydrodynamic loop” predicted by an unsteady flow model. This difference is normally attributed to poorly quantified changes in channel conveyance driven by sedimentation processes.

Chapter 6

One-Dimensional vs Two-Dimensional Modeling

The question of 1D versus 2D hydraulic modeling is a much tougher question than steady versus unsteady flow. There are definitely some areas where 2D modeling can produce better results than 1D modeling, and there are also situations in which 1D modeling can produce results that are just as good as 2D models with less effort and computational requirements. Unfortunately, there is a very large range of situations that fall into a gray area, and one could list the positive and negative aspects of both methodologies for specific applications.

This chapter discusses the differences between one-dimensional (1D) and two-dimensional (2D), steady and unsteady flow, hydraulic modeling. Specifically this chapter describes the definition of 1D and 2D hydraulic modeling and assumptions used in each; the equations used for 1D and 2D unsteady flow modeling; computational differences; applications examples; model development; model calibration; modeler, knowledge, skills, and abilities; and a summary of 1D and 2D modeling advantages and disadvantages.

Definitions

In general, almost all fluid movement is three dimensional. However, the equations of water motion are often derived in both one and two-dimensional forms, for a wide range of practical applications.

When the equations of motion are derived in a one-dimensional form, it is under the assumption that the forces acting on a body of water are predominant in one direction, x, along the river channel centerline. This assumes that vertical and lateral forces acting on the water body are small in comparison to the x direction, and therefore can be assumed to be negligible. So the one-dimensional form of the equations of motion only account for forces in the x direction.

When the equations are derived in a two-dimensional form, it is under the assumption that the forces acting on a body of water are predominant either in the x (along the river channel centerline) and the y (laterally across the channel or floodplain), which is the most common form for 2D modelling, or in the x and the z (vertically across the depth). For the form of the equations written in terms of x and y, vertical averaging (z, or vertical forces) assumes that the vertical forces acting on a body are small in comparison to the horizontal forces (x, y), and are therefore considered to be negligible. For the form of the equations written in terms of x and z, lateral averaging (y, or lateral forces) assumes that the lateral forces are negligible.

One and Two-Dimensional Equations

The physical laws which govern the flow of water in a stream are: (1) the principle of conservation of mass (continuity), and (2) the principle of conservation of momentum. These laws are expressed mathematically in the form of partial differential equations, which will hereafter be referred to as the continuity and momentum equations. In the derivation of both the one-dimensional (1D) and two-dimensional (2D) unsteady flow equations (often called the shallow water equations), there are several assumptions made about the flow:

1. Water is an incompressible fluid.
2. The pressure distribution is considered to be hydrostatic.
3. The vertical acceleration of the water is considered to be negligible.
4. The bed slope is considered to be mild (less than a 1:10 slope).
5. The effects of boundary friction can be taken into account with flow resistance laws derived for steady flows (i.e., Manning's equation).
6. The Boussinesq approximation is valid (ignoring forces caused by differences in water density).

One-Dimensional Equations (1D)

The one-dimensional continuity and momentum equations can be written in partial differential equation form, with respect to Depth (h) and Velocity (u), and are commonly shown in hydraulic text books as follows:

Continuity Equation:

$$\frac{\partial h}{\partial t} + \frac{\partial (hu)}{\partial x} - q = 0$$

Momentum Equation:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + g\left(\frac{\partial h}{\partial x} - S_0 + S_f + S_h\right) = 0$$

where:

u	= velocity in the x direction
h	= depth of water
g	= gravity
t	= time
x	= distance in the direction of flow (x plane)
q	= Lateral inflow term (source/sink)
S_0	= Bed slope
S_f	= Friction slope, from Manning's equation.
S_h	= added force term (additional minor losses)

Figure 6-1 provides a diagram defining the symbols used in the 1D and 2D equations for motion.

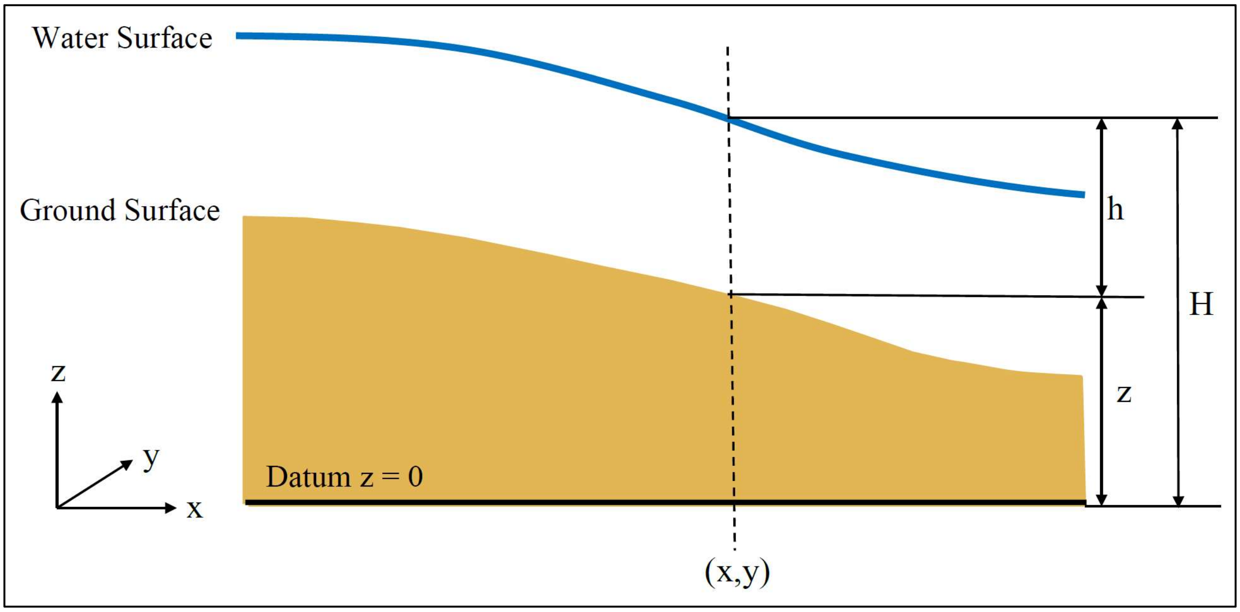

Figure 6-1. Definition of Symbols used in the 1D and 2D equations of motion.

Two-Dimensional Equations (2D)

The two-dimensional continuity and momentum equation can be written in partial differential equation form, with respect to depth (h) and velocity (u, v, U, V), and are commonly shown in hydraulic text books as follows:

Vertically Averaged Continuity Equation:

$$\frac{\partial H}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} - q = 0$$

Vertically Averaged Momentum Equations:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -g\frac{\partial H}{\partial x} + v_t\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) - c_f u + fv$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -g\frac{\partial H}{\partial y} + v_t\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) - c_f v - fu$$

where:

v = velocity in the y direction
y = distance in the lateral direction (y plane)
H = water surface elevation (z + Depth)
v_t = horizontal eddy viscosity coefficient
c_f = bottom friction coefficient
f = Coriolis parameter

Laterally Averaged Continuity Equation:

$$\frac{\partial UB}{\partial x} + \frac{\partial WB}{\partial z} - qB = 0$$

Laterally Averaged Momentum Equations:

$$\frac{\partial \text{UB}}{\partial \text{t}} + U\frac{\partial \text{UB}}{\partial x} + W\frac{\partial U\text{B}}{\partial z} = -g\frac{\partial BH}{\partial x} + \frac{B}{\rho}\frac{\partial P}{\partial x} + \frac{1}{\rho}\frac{\partial B\tau_{xx}}{\partial x} + \frac{1}{\rho}\frac{\partial B\tau_{xz}}{\partial z}$$

$$\frac{1}{\rho}\frac{\partial P}{\partial z} = -g\frac{\partial H}{\partial z}$$

where:

v	= velocity in the y direction
y	= distance in the lateral direction (y plane)
H	= water surface elevation (z + Depth)
v_t	= horizontal eddy viscosity coefficient
c_f	= bottom friction coefficient
f	= Coriolis parameter
U	= laterally averaged velocity in x direction
W	= laterally averaged velocity in z direction
B	= width
P	= laterally averaged pressure
τ_{xx}, τ_{xz}	= turbulent stresses in the xx, and xz directions, respectively
q	= lateral inflow per unit volume

As displayed above, the 1D and 2D shallow water equations are very similar, with the main differences being: the 1D equations are only derived for forces acting in the x direction (Figure 6-1), while the 2D equations account for forces acting in either the x and y direction, or the x and z direction (Figure 6-1); the 1D equations use an added force term to describe additional forces due to severe contractions and expansion (based on an empirical coefficient), while the 2D equations solve for this directly with the inclusion of the Eddy Viscosity/Turbulent stress terms; and the 2D equations have an added term to account for the rotation of the earth (Coriolis), which cannot be included in the 1D approach. In general practice Coriolis effects are neglected in the laterally averaged 2D equations, unless you are modeling areas closer to the north and south pole, where the Coriolis affect is more predominant.

Terms in the presented equations can be modified to account for density differences, or additional terms can be added for other forces, such as wind and mud/debris flows. Note that the presence of density differences does not violate the Bousinnesq approximation.

Diffusion Wave Form of the Momentum Equation

Simplifications of the 1D and 2D forms of the equations can be made, with the most common being the Diffusion Wave form of the equations. The Diffusion Wave approximation is often used in both 1D and 2D solution forms. To get to the diffusion form of the equations, simply drop the acceleration terms (changes in velocity with respect to time and space) in the momentum equations. This form of the momentum equation only contains gravity, friction, and hydrostatic pressure forces.

The diffusion form of the momentum equation can be combined with the continuity equation, and written in terms of solving for only the water surface elevation. This makes the equation easier to solve, generally more stable for a wide range of problems, and will require much less computational time. However, without the acceleration terms in the equations, the diffusion wave equations are less accurate than the full equations, and they are less applicable to the full range of problems that the modeler may need to solve. This will be discussed further in this document.

Henceforth, 2D will be assumed to imply 2D-Vertically Averaged Shallow Water Equations unless otherwise specified.

Computational Differences

In order to better understand the differences between 1D and 2D unsteady flow modeling, the modeler should be aware of all of the computational differences between the two approaches. The following is a description of the major computational differences between 1D and 2D unsteady flow routing.

Water Surface and Velocities

The most obvious difference between 1D and 2D models is that 1D models only compute a single water surface elevation at each cross section, while 2D models compute a unique water surface for every cell/face in the model (see example in Figure 6-2). Additionally, velocities in a 1D model are computed as averaged velocities (Vertically and horizontally) for the main channel, left overbank, and right overbank within each cross section (see example in Figure 6-3). Some 1D models, like HEC-RAS, compute 1D velocities between Manning's n value break points for the overbank area, and can additionally post process results into more spatial velocities. However, those velocities are based on the assumptions that the flow is perpendicular to the cross section, and the flow distribution is only a function of the cross section's conveyance distribution.

2D models compute averaged velocities (vertically and horizontally) for each cell face. However, the detailed of the water surfaces and the velocities in a 2D model depends on the number of cells that a modeler uses across the main channel and the floodplains (see example in Figure 6-4).

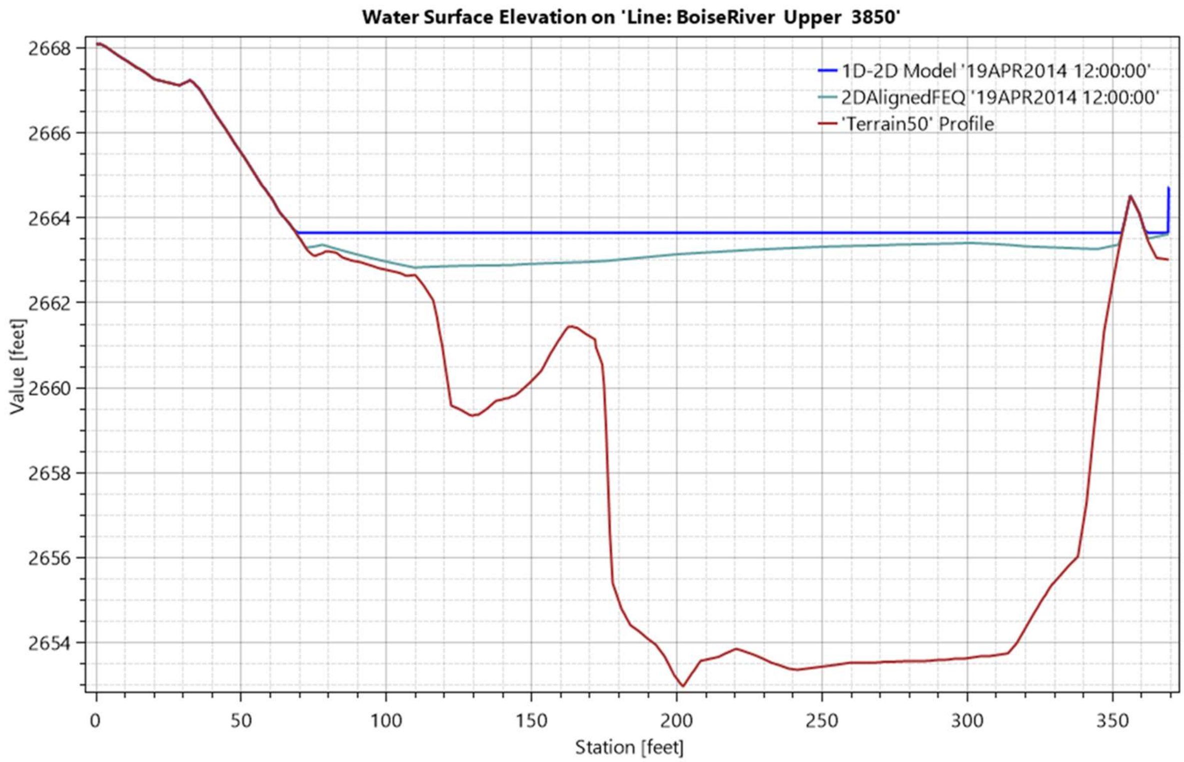

Figure 6-2. Example Water Surface Plot for a 1D and 2D model solution.

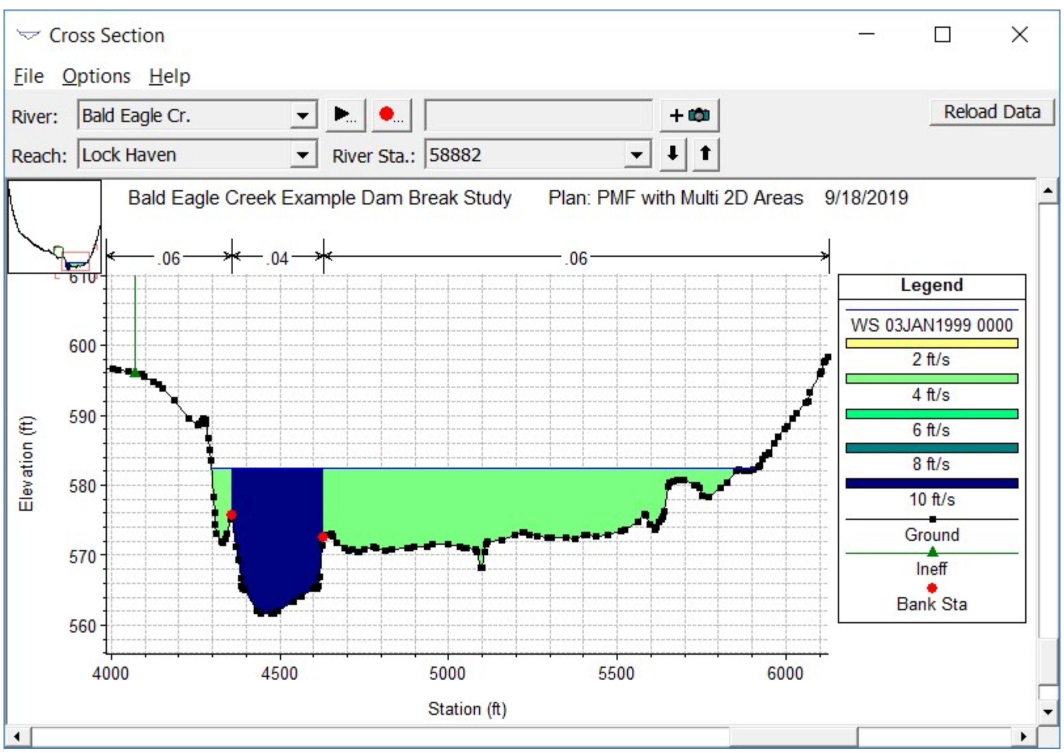

Figure 6-3. Example velocity output from a 1D Model.

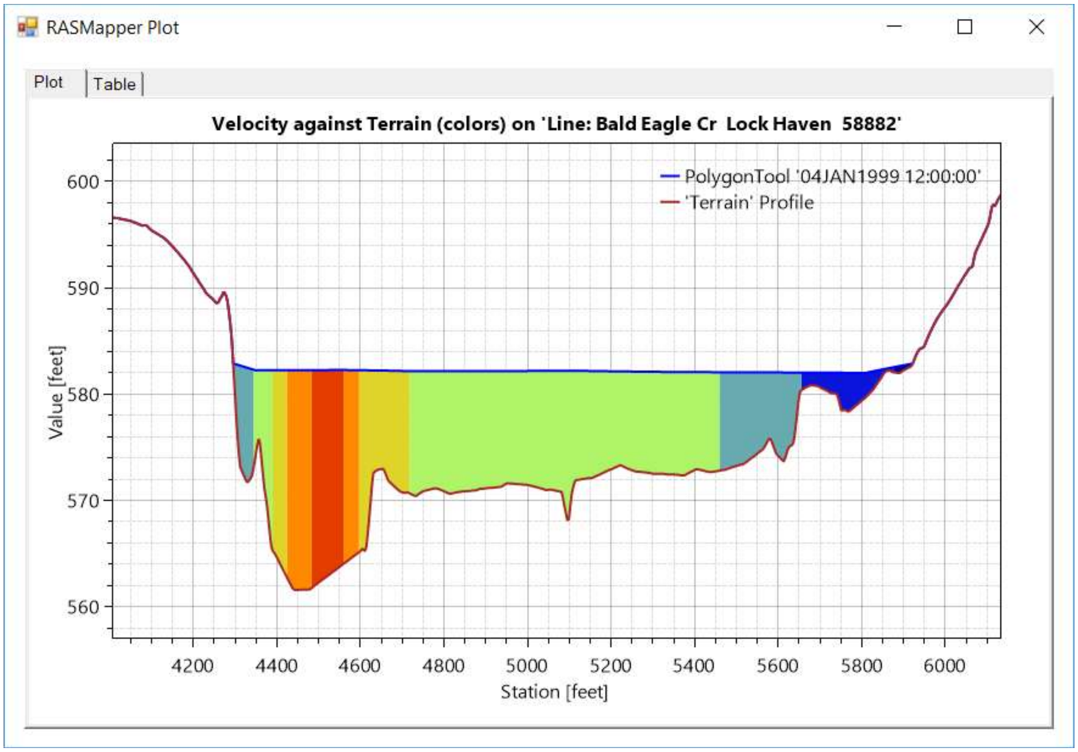

Figure 6-4. Example velocity output at a cross section from a 2D model.

Friction Losses

Friction losses are computed for 1D models by multiplying an averaged friction slope (S_f) by the length between cross sections. The friction slope is computed at each cross section with Manning's equation. Different averaging techniques can be used to compute the average friction slope between cross sections. Additionally, the distance between cross sections is flow weighted based on the flow in the left overbank, main channel, and right overbank, and their corresponding reach lengths. 2D models also compute a friction slope at each face of the cells, however, the friction slope is generally not averaged over the cell, as the direction of the flow is in two dimensions. The friction slope is used to compute a frictional force at each face, and is included as a component in the solution of the two dimensional momentum equation.

Conveyance Calculations

For 1D models, conveyance is calculated for the main channel of the cross section as a separate flow area. This means the entire area, wetted perimeter, and roughness are used to calculate a single conveyance for the main channel at each water surface elevation. The conveyance in the overbanks is also split up into separate flow areas based on breaks in roughness. So, an overbank area will be treated as a separate conveyance area if it has a single Manning's n value, or it can be treated as several conveyance areas if it has multiple Manning's n values. An example of a 1D cross section broken into separate conveyance areas is shown in Figure 6-5. As shown, the left overbank is broken into two conveyance areas, even though they have the same Manning's n value. The conveyance is broken up this way because Manning's equation was developed for separate flow areas (originally channels only). Given the variability of the depth and velocity, and the relationship between area and wetted perimeter in in Manning's equation, treating the left overbank as two separate flow areas is more appropriate.

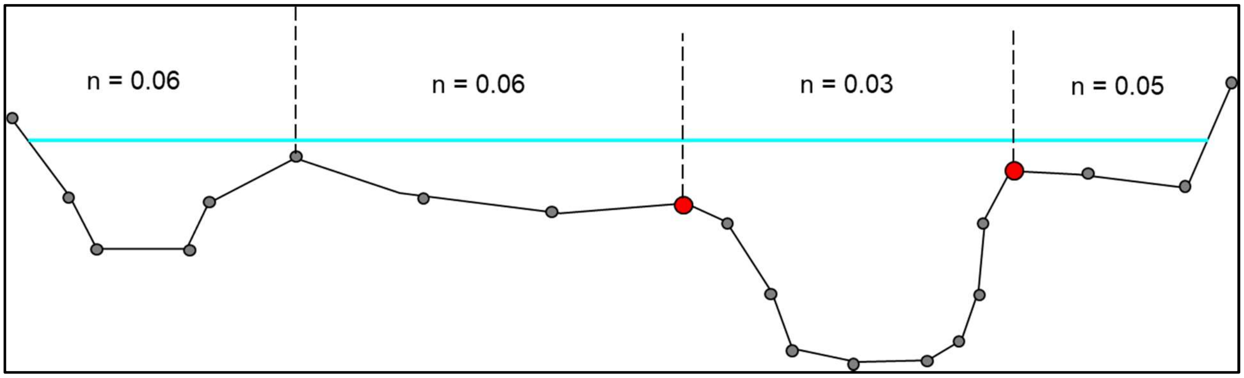

Figure 6-5. Example of Separate Conveyance Areas for a 1D Cross Section.

2D models compute conveyance separately for every face of every cell. This can lead to differences in results between 1D and 2D models, even for just main channel flow only. For example, let's say we have a simple trapezoidal channel. A 1D model will treat the channel as a single flow area, and compute conveyance using the total area and wetted perimeter for every water surface elevation. For a 2D model it will depend on how many cells are being used to model the main channel. If a single cell is being used, then the entire area and wetted perimeter will be used to compute conveyance, just like in a 1D model. However, if several cells are being used to model the main channel, then conveyance is computed separately for each face that crosses the channel. This approach will produce a different amount of conveyance for a given water surface. In general, breaking the channel up into pieces will produce a higher amount of conveyance for a given water surface. The net effect of this is that when a channel is broken into several pieces for computing the conveyance, the computed water surface will be lower due to the perceived increase in efficiency of the higher channel conveyance.

An example trapezoidal channel is shown in Figure 6-6. This channel was modeled with 1D cross sections, and also with 2D cells. For the 2D modeling, three different 2D models were created (using HEC-RAS). One 2D model was created as a single cell model across the entire channel, another was created with 4 cells across the channel (shown in Figure 6-6), and a third was created with 10 cells across the channel. All models used the same downstream boundary condition, which was a single valued rating curve. The results of this experiment are shown in Figure 6-7. The 1D model and the 2D model, using a single cell for the channel, give basically the same results (the higher computed water surface elevations). The 2D models using 4 and 10 cells across the channel also gave about the same results, but lower than the 1D model and single cell approach. This is due to the fact, that if you break the channel into pieces, and separately compute conveyance for those pieces, you do not get the same conveyance as when computing the conveyance as a single flow area. Conveyance is highly nonlinear with respect to the relationship between area and wetted perimeter. The greatest differences will occur where there are steep banks that get computed separately from the main portion of the channel. In the example shown in Figure 6-6 (4 cells), the conveyance that is computed for the cells that represent the banks is very low, due to the large amount of wetted perimeter to flow area ratio.

The two cells in the middle have much smaller wetted perimeter to flow area ration, and thus much higher conveyance. The overall conveyance for the entire channel ends up being greater than if you computed it as a single flow area, and the net results is a lower water surface.

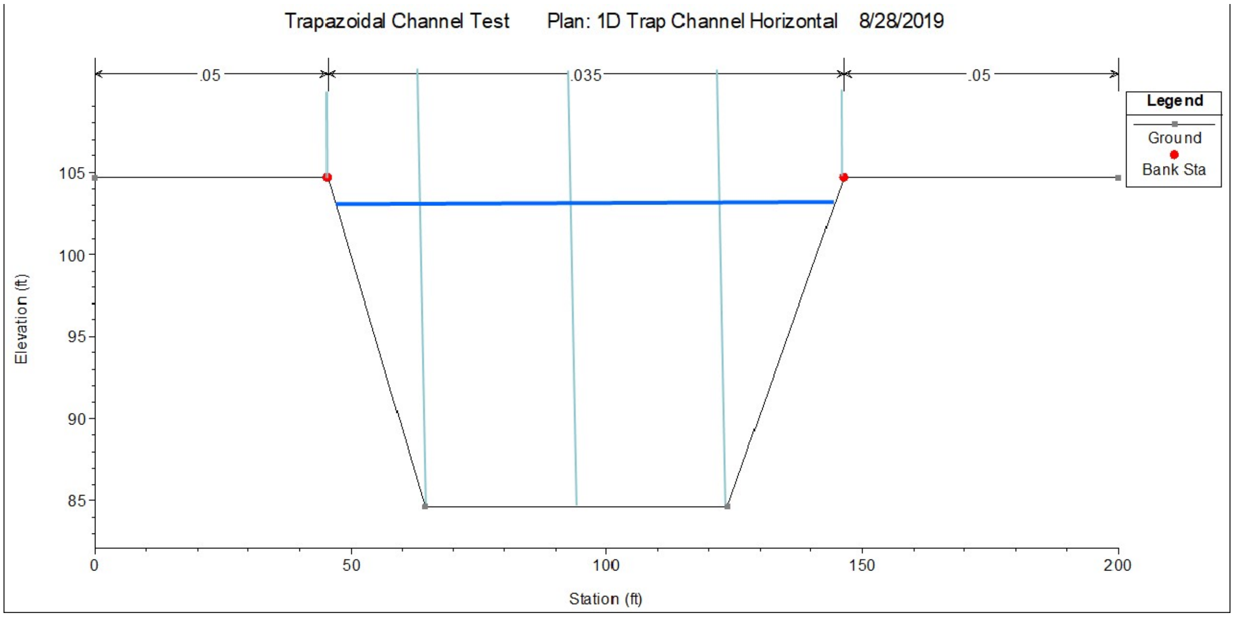

Figure 6-6. Example Trapezoidal Channel for 1D and 2D Conveyance Computations.

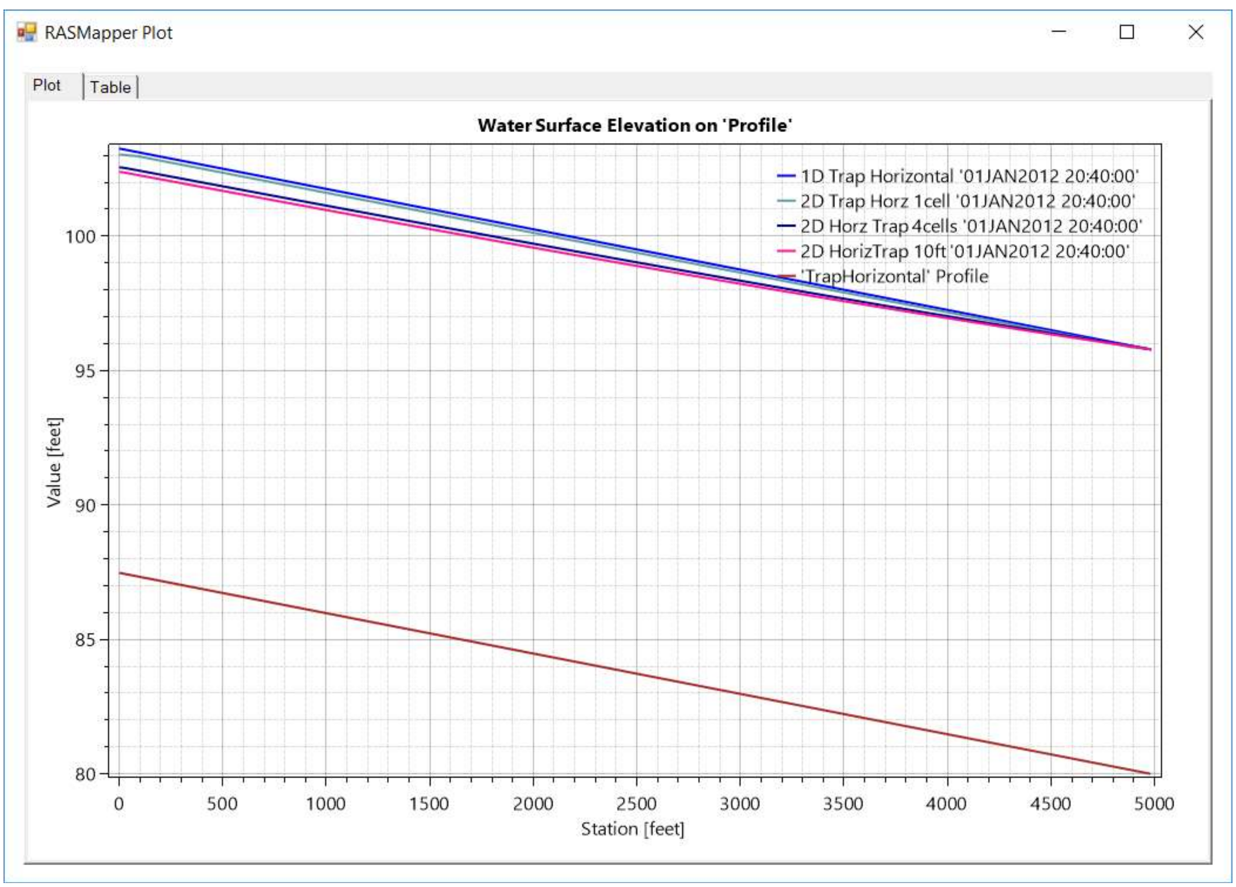

Figure 6-7. Results for Computing Conveyance with 1D and 2D models of varying cell size.

Physically what will really happen is that the slower water in the area of the banks will slow down the water going towards the middle of the channel, and the faster water in the middle of the channel will affect the velocity of the water over the channel banks. This phenomenon is handled in 2D models through the use of turbulence modeling. The water to water shear force between two adjacent cells can be accounted for directly in 2D models with turbulence modeling. Shown in Figure 6-8, is the same experiment, however, turbulence modeling was turned on for all of the 2D modeling approaches. As shown, now all the models, 1D, 2D single cell, 2D four cells, and 2D ten cells, are all getting about the same water surface elevations. However, a user specified turbulence coefficient had to be entered for the 2D models. This coefficient was calibrated to give the same results as the 1D model.

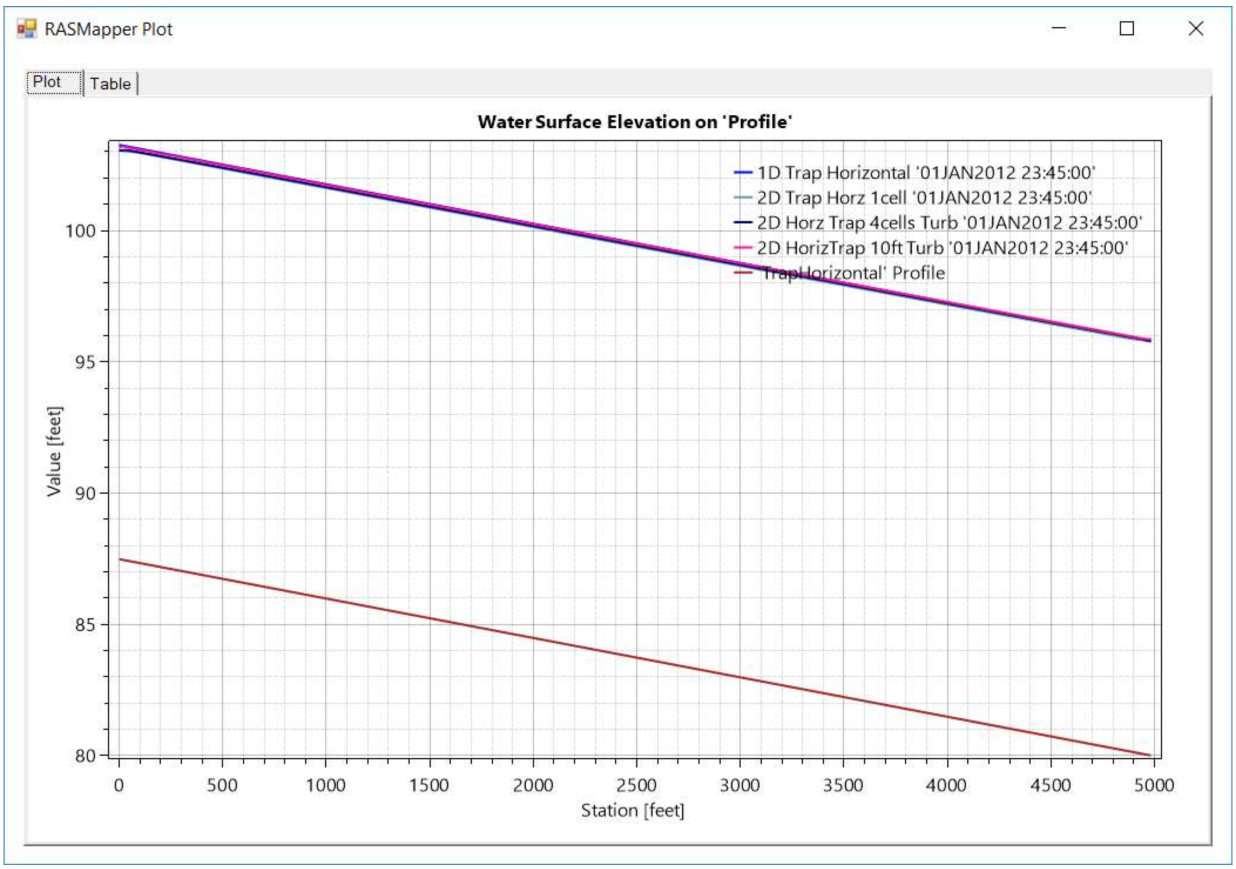

Figure 6-8. Example Results for a Trapezoidal Channel Modeled in 1D and 2D with Turbulence Modeling Turned on.

Contraction and Expansion Losses

Contractions and expansions of flow are inherently a three dimensional flow phenomenon. 1D models handle the force/energy losses at contractions and expansions using empirical coefficients multiplied by a change in velocity head (steady and unsteady flow 1D models). Additionally, 1D unsteady flow models compute spatial acceleration forces and pressure differential forces through the contraction and expansion of the flow. However, the spatial acceleration and pressure forces are only computed in one direction.

2D models capture the forces due to contractions and expansions with two dimensional pressure forces and spatial acceleration terms. Additional forces are captured with turbulence modeling to account for all of the force/energy losses associated with a flow contraction/expansion. As mentioned previously, turbulence modeling requires the user to enter and calibrate turbulence coefficients.

Ineffective Flow Areas

1D models will use the entire cross section to represent actively moving water. However, portions of a cross section will often have water that is not conveying (no velocity) in the downstream direction. Because of non-conveying areas, 1D modeling requires the use of "Ineffective Flow" areas to model the portions of the cross section that is wet but not producing any downstream conveyance. This requires a lot of judgment from the user in order to place these ineffective flow areas correctly within the cross section. Additionally, the ineffective flow areas may need to be turned off when the water surface gets high enough, due to the fact that the area will now convey water in the downstream direction for the higher water surface elevation. So an additional parameter, called a "trigger elevation" must also be entered for each ineffective flow area. This trigger elevation also requires user judgment.

2D models do not require the user to define any ineffective flow areas. Ineffective flow areas, or recirculation zones, are automatically computed based on the 2D equations. However, turbulence modeling and turbulence coefficients can affect the size of recirculation zones and the velocity of the water near these zones.

Application Examples

As mentioned previously, there are definitely modeling application areas in which 2D modeling can produce better results than 1D modeling. Here are some application examples in which 2D modeling will always give better results than 1D modeling:

- Levee overtopping and/or breaching where water can go in many directions. If a levee interior area has a slope to it, water will travel overland in potentially many directions before it finds its way to the lowest point of the protected area, and then the water will begin to pond and potentially overtop and/or breach the levee on the lower end of the system. However, if a protected area is small, and ultimately the whole area will fill to a level pool, then 1D modeling is fine for predicting the final water surface and extent of the inundation. An example of a leveed system with a breach is shown in Figure 6-9. As shown, water is going in many directions, down streets, across roads, and into low lying areas.

- Areas and/or events in which the flow path of the water is not completely known. For example, if a dambreak analysis is to be performed, but the downstream area is not confined or is very flat, then exact knowledge of where the water will go may not be possible. 1D models require knowledge of the flow path for all events before the model geometry can be defined. 2D models do not have this requirement, as flow can go in any direction within the 2D modeling domain. Additionally, because a dambreak is highly nonlinear, the full form of the equations should be used (including the acceleration terms), as a dambreak has extremely rapid changes in depth and velocity with respect to space and time. An example of a dambreak, in which the flow goes out into a very flat area, and then travels in many directions is shown in Figure 6-10.

- Very wide and flat flood plains, such that when the flows goes out into the overbank area, the water will take multiple flow paths and have varying water surface elevations and velocities in multiple directions.

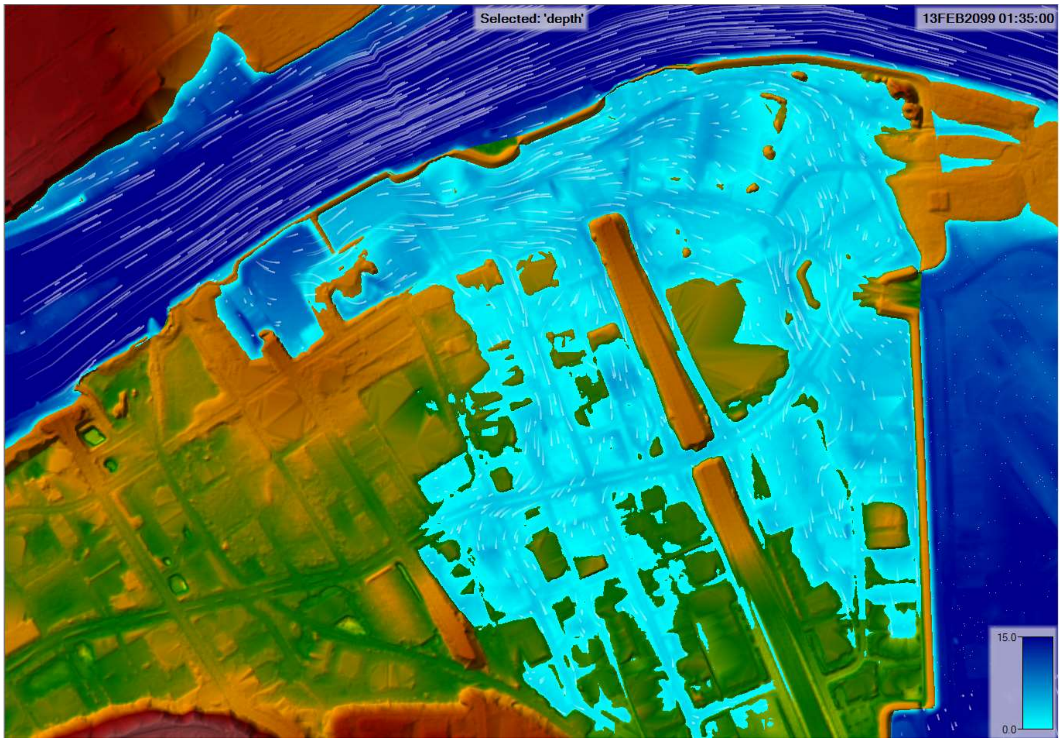

Figure 6-9. Example of a leveed system breach with water going in many directions.

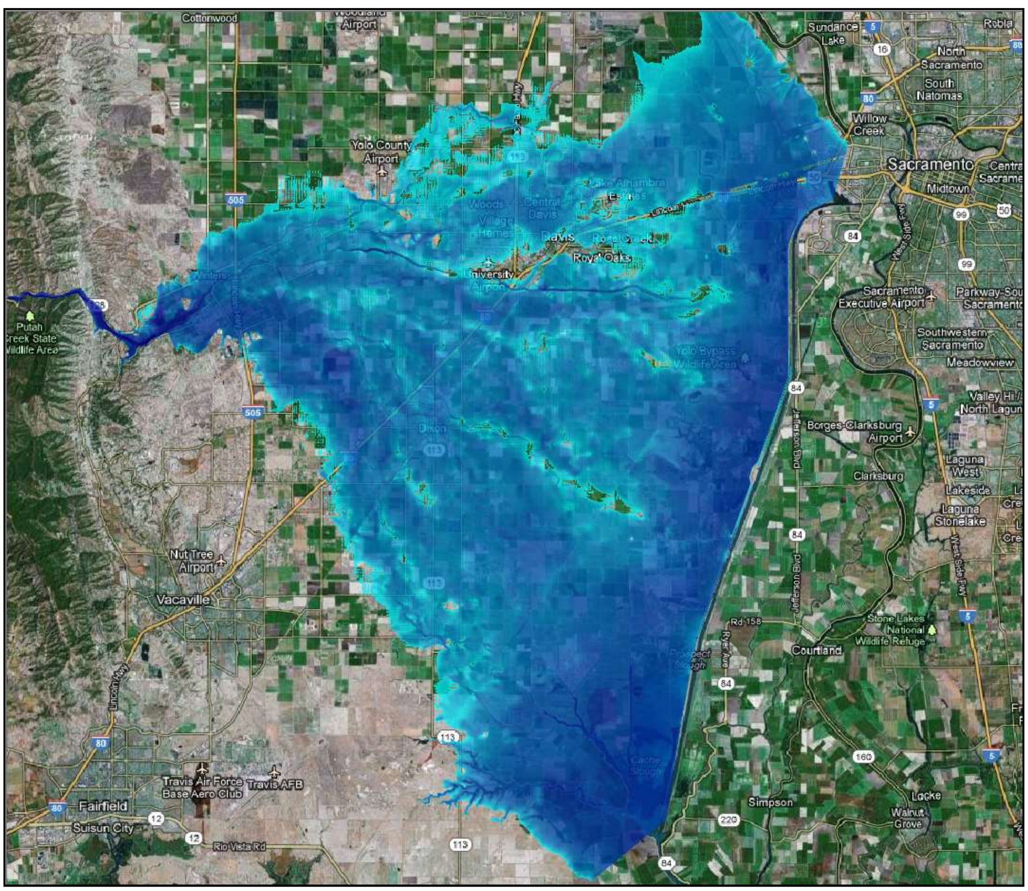

Figure 6-10. Example Dambreak that goes out into an extremely flat area and spreads out. Water depths shown in shades of blue (dark blue indicates greater water depth).

- Bays and estuaries – Within bays and estuaries, flow will continuously go in multiple directions due to tidal fluctuations and river flows coming into the bay/estuary at multiple locations and times. Displayed in Figure 6-11 is the Columbia River Estuary. As shown, water will move in many directions within the estuary. In order to get accurate velocities and flow directions, 2D or even 3D modeling would be necessary. Tidally driven rivers also require using the full form of the equations, including the acceleration terms. Wave propagation in general cannot be solved accurately without the acceleration terms in the momentum equation.

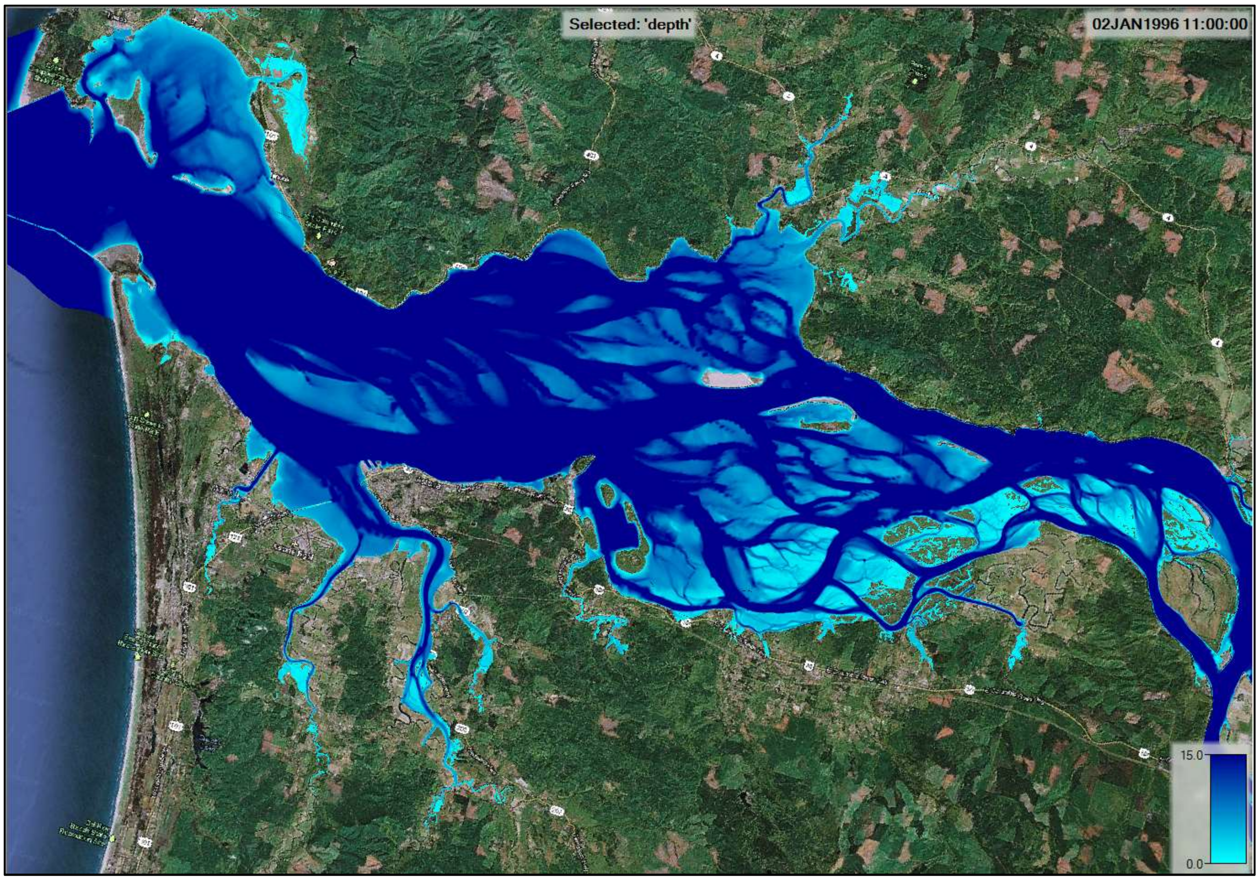

Figure 6-11. Lower Columbia River Estuary with water depths shown in shades of blue.

- Highly braided streams – In a stream that is highly braided, the channel splits and combines numerous times. This type of system is very difficult to model accurately with a 1D modeling approach.

- Alluvial Fans – It is debatable that any rigid boundary numerical model can capture a flood event accurately on an alluvial fan, due to the episodic nature of flow evolutions that can change the terrain surface, conveyance, and the whole direction of the channels during the event. In general, water will split into many directions and spread out over the alluvial fan. This would be extremely difficult to model in 1D.

- Flow around abrupt bends, in which a significant amount of super elevation occurs. For example, Figure 6-12 depicts a 2D model of a concrete rectangular channel going around a 180 degree bend. A 2D model was used in order to capture the super elevation that occurs around the bend. The velocities that occur at this type of location are very complex in all three dimensions, and would most likely have spiral flow patterns. If a detailed velocity distribution through the bend was needed, then a 3D model or physical model would be better suited for this type of problem. In other words, this is another area where the full 2D and 3D equations are required, as the acceleration terms are used to model the super elevation around the outside bend. Wave run-up in general can only be computed with the inclusion of the acceleration terms, and this includes the rise in the water surface around an object or a wall.

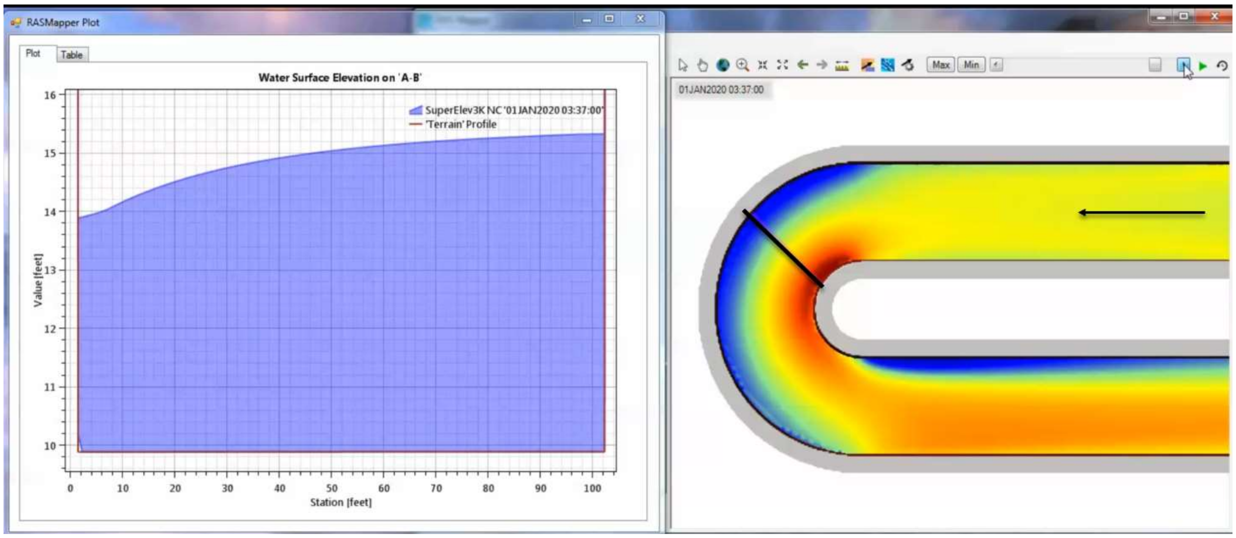

Figure 6-12. Example of super elevation of the water surface around a sharp bend.

- Applications where vertical velocity gradients exist. These problems might require the application of a 2D model or a 3D model. An example of a 2D laterally averaged model depicting the vertical velocity gradient is shown in Figure 6-13.

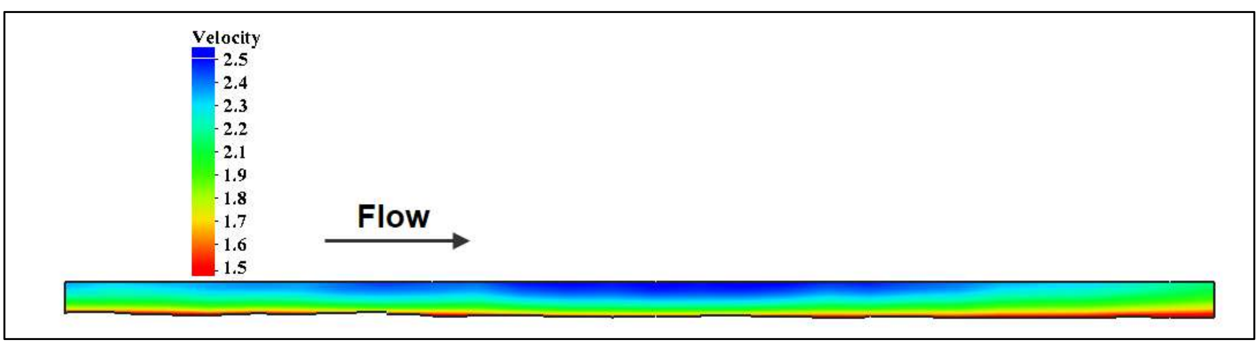

Figure 6-13. Detailed 2D-Laterally Averaged model of vertical velocity gradients. The directions shown are x and z.

- Applications where it is very important to obtain detailed velocities for the hydraulics of flow over or around an object, such as a bridge abutment or bridge piers. This type of application also requires the full form of the equations, as the diffusion wave form cannot account for wave run-up in front of the object, or the rapid deceleration and acceleration in front of and around the object. An example of a 2D model depicting detailed velocities of flow going around piers is shown in Figure 6-14. However, flow around obstructions is implicitly 3D in nature; therefore, if detailed behavior of velocity, scour, or sediment is required then a non-hydrostatic 3D model is necessary.

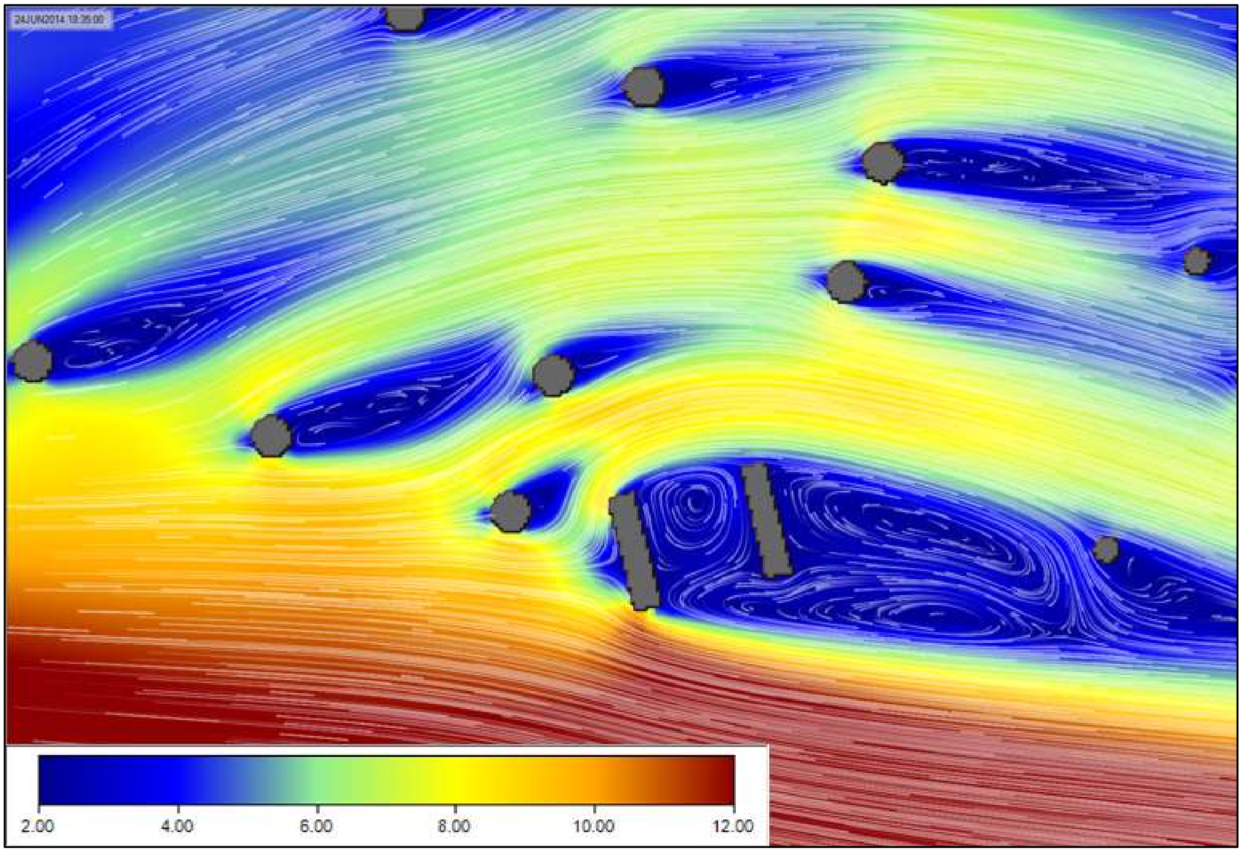

Figure 6-14. Detailed 2D model velocity plot of flow going around piers from a railroad station platform.

The following are areas in which 1D modeling can potentially produce results as good as 2D modeling (from the perspective of computed water surface elevations, and flow/stage hydrographs), with less effort (from a model development, calibration, and application viewpoint, as well as a computational time viewpoint):

- Rivers and floodplains in which the dominant flow directions and forces follow the general flow path of the river, and where the channel and floodplains are directly connected and the water surface along the cross sections are relatively constant. This situation potentially covers many river systems, but it is obviously debatable as to the significance that lateral and vertical velocities and forces impact the computed water surface elevations and the resulting flood inundation boundary. An example of this type of river is shown in Figure 6-15 which displays the confluence of the Allegheny and the Monongahela rivers at Pittsburgh, PA. In this example situation, the two rivers flow through rolling hills and small mountain ranges. The rivers are cut deep into the topography, and there is very little overbank area for water to spread out when flow goes out of the channel. The flow lines follow the stream centerline very closely, and the water is therefore highly one-dimensional.

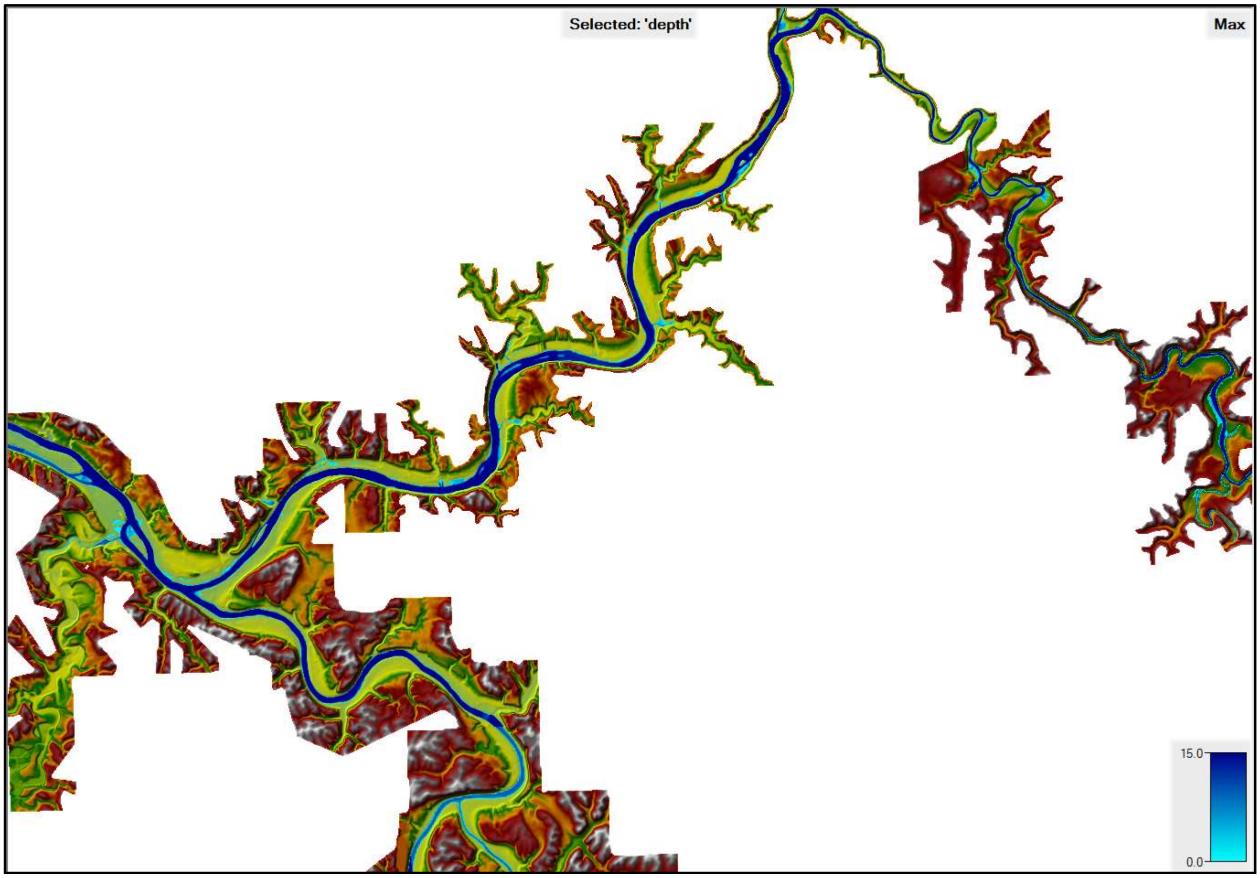

Figure 6-15. Example of a highly one-dimensional flowing river system (Allegheny - Monongahela Rivers, confluence at Pittsburgh, PA). Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth).

- Steep streams that are highly gravity driven and have small overbank areas. Consequently, this situation is also an area where the diffusion wave form of the equations will produce similar results to the full form of the equations. The reason the equations produce similar results is due to the fact that the magnitude of the acceleration terms is very small in comparison to the gravity and frictional forces that dominate the movement of water in steep streams.

- River systems that contain many bridges/culvert crossings, weirs, dams and other gated structures, levees, pump stations, etc., that impact the computed stages and flows within the river system. For these types of river systems the current state of the art in 1D models is still ahead of the 2D models; however, this will change in time as 2D modeling software improves. Moreover, this is also a good location to perform combined 1D and 2D modeling for the river system. 1D modeling for the main channel and hydraulic structures, and then 2D modeling for the overbank areas and floodplains. Shown in Figure 6-16 is an example of a combined 1D/2D model of the Truckee River system that flows through Reno, NV and then goes through the Truckee Meadows area. The main river is steep and it has many bridges crossing the stream. While the flow in the main river is highly one dimensional, the flow in the overbank areas is highly two dimensional. The modeling of the Truckee main river is complicated due to the numerous bridges and weirs that cross the stream, intermittent levees that get overtopped in both directions and significant differences in water surface between the channel and overbanks. Subsequently, this is an ideal example situation to use 1D modeling for the main river and 2D modeling for all of the overbank areas.

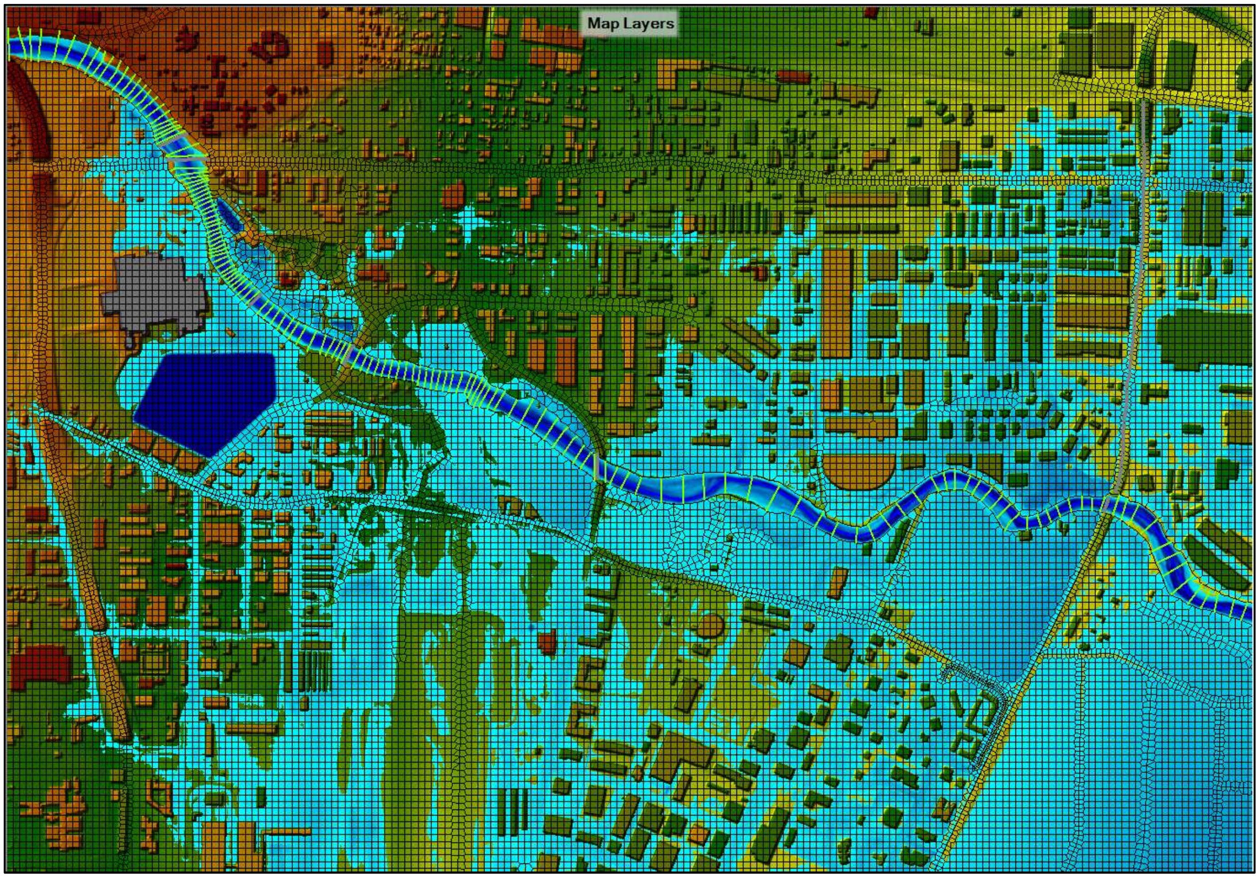

Figure 6-16. Example 1D/2D model of the Truckee River near Reno, NV. Green to red color indicates the terrain (low to high elevation) and the blues indicate water depth (dark blue indicates greater depth).

- Real time forecasting applications which model medium to large river systems (100 or more miles of the river system) and run longer time period forecasts (i.e., weeks to months). Even with the tremendous advancements in multi-processor computing, and GPU (Graphics Processor Units) computing, there are still significant spatial and simulation time limitations on what/when we can effectively use 2D models for in the real time forecasting. Simulation run times for 2D modeling will continue to improve over time.

- Areas in which the basic data does not support the potential gain of using a 2D model. If detailed overbank terrain and channel bathymetry do not exist, or if there is only detailed cross sections at surveyed locations, many of the benefits of the 2D model will not be realized due to the poor accuracy of the terrain data.

Model Development

While the development of a hydraulic model is similar, regardless of whether it is 1D or 2D, there are also many significant differences in the model development process. This section discusses the model development process, and highlights differences between the two approaches. Additionally, advantages and disadvantages of the 1D and 2D modeling approaches are discussed.

Terrain Data

As discussed previously, all hydraulic model development requires terrain data. Specifically, 2D models require that a complete terrain model is built that includes topography for the entire modeling domain, including the channel. On the other hand, for 1D models it is common to use either surveyed cross sections for the channel and floodplain areas, or a terrain model for the overbank areas and surveyed cross sections of the channel area. For more details on terrain needs for 1D and 2D modeling approaches, see Chapter 4.

Model Discretization

Before laying out a 1D model, the flow path of the water must be known for all events to be modeled. Additionally, if the flow paths/direction change with different events, this would require more than one geometric representation of the system. On the other hand, 2D models do not require knowledge of the exact flow paths of the water. They do require knowledge of the extent of flooding (which is also true for 1D models).

Once the model extents are defined, then the channel and floodplain area are then defined with either cross sections (1D models) or a computational mesh (2D models). For the 1D modeling approach, level of detailed is defined by how many cross sections are used, and ensuring that cross sections are placed at the correct locations. In general for 1D modeling, cross sections need to be spatially located to accurately capture the terrain, friction losses, and contractions and expansion losses. However, the number of cross sections needed for 1D modeling is highly dependent on the slope of the stream (i.e., steeper streams require more cross sections at shorter distances apart).

Additionally, cross sections must be placed at breaks in grade (slope changes); around hydraulic structures; at significant changes in roughness; junctions and flow splits; and locations where flow will change. In 1D modeling, in order to get an accurate representation of the true flow area, modelers must layout cross sections perpendicular to the flow, as well as define ineffective flow areas and blocked obstructions to the flow within each cross section. This requires knowledge of the flow paths and how much of the cross section will be active for the range of events.

Development of a 2D model consists of defining a base mesh cell/element size for the problem, then refining/coarsening the mesh in the appropriate areas. In general, smaller cell sizes are required for locations where the terrain is changing significantly and where the water surface elevation and/or the velocity are changing. This requirement is due to the fact that, dependent on the model used, a single water surface is computed for each cell, and an average velocity is computed for each cell face. Depending on the desired level of detail of the water surfaces and velocities, more cells will be required in areas where more change is occurring.

The development of a good 2D mesh will have additional requirements beyond what is listed above. Refinement and alignment of the cells contained with the main channel is often required to get better results. This will include aligning the faces of the cell along the high ground of the main channel bank stations, in order to ensure the flow stays in the channel until bank elevation is exceeded. Aligning the cells with the flow will also reduce any numerical diffusion that may be present in the solution scheme being used to solve the equations. And finally, the number of cells across the channel will affect the details of the computed velocity distribution. Additionally, most 2D models computationally represent each cell as a single elevation, and each face as a single line. Some models use triangles for each cell, which represents the land surface with 3 points. These types of model will require more cells across the channel in order to accurately represent the channel area and wetted perimeter correctly. Some 2D models use subgrid technology (HEC-RAS), which represents the cell as a detailed elevation – volume relationship and each face as a detailed cross section, based on the underlying terrain. Subgrid models will require far fewer cells in order to represent the channel area and wetted perimeter correctly. These types of models can model a channel accurately with just a few cells for the entire channel (3 to 4 cells). However, if the user wants a detailed velocity distribution within the channel, then more cells will be required, as each cell/face only computes a single average velocity. In general, to get a good velocity distribution across the channel you will need to use at least 5 to 10 cells across the channel, depending on how detailed you want the velocity to be discretized.

Additional requirements for creating good 2D meshes will include aligning cells along the tops of roads, levees, floodwalls, and any other barriers to flow, even if it is a natural terrain barrier.

In 2D modeling there is no requirement to define ineffective flow areas, as the 2D approach can automatically identify areas of active flow and eddy zones. Some 2D models do not require the mesh cells to be aligned with the flow direction. However, aligning the mesh cells for the main channel flow will allow the model to produce less numerical error and a less diffusion of the water surface elevation (attenuation of the peak). An example of a 2D mesh with the main channel cells smaller than the overbank cells, and the main channel cell/elements aligned with the flow is shown in Figure 6-17.

If detailed velocity behavior descriptions are required around structures such as bridge piers, bendway weirs, or other structures, sufficient resolution should be provided around them to allow the 2D model to accurately capture the behavior of interest. An example of a 2D detailed mesh of increased resolution around a structure is shown in Figure 6-18.

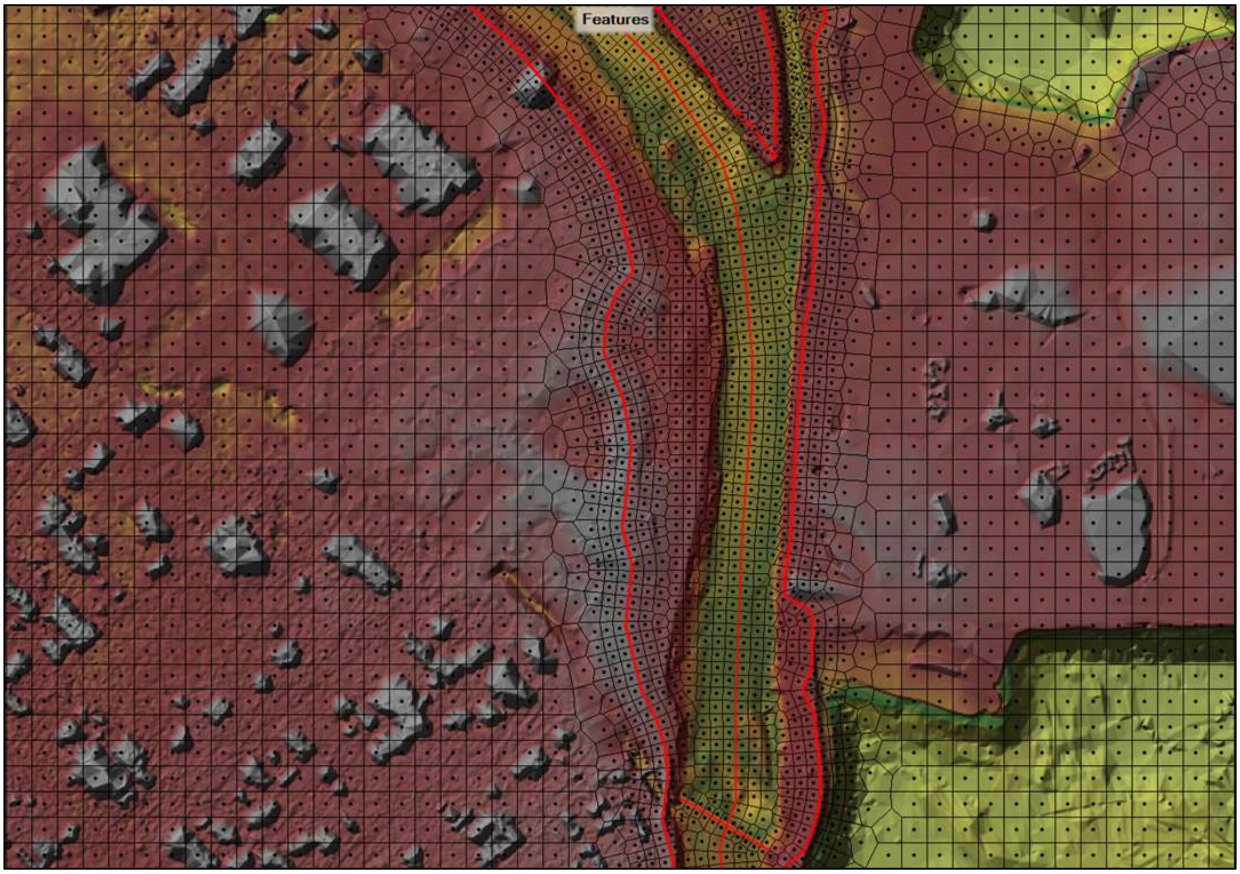

Figure 6-17. Example 2D mesh with a detailed mesh of the main channel, with grids aligned to the flow. Green to red color indicates the terrain (low to high elevation).

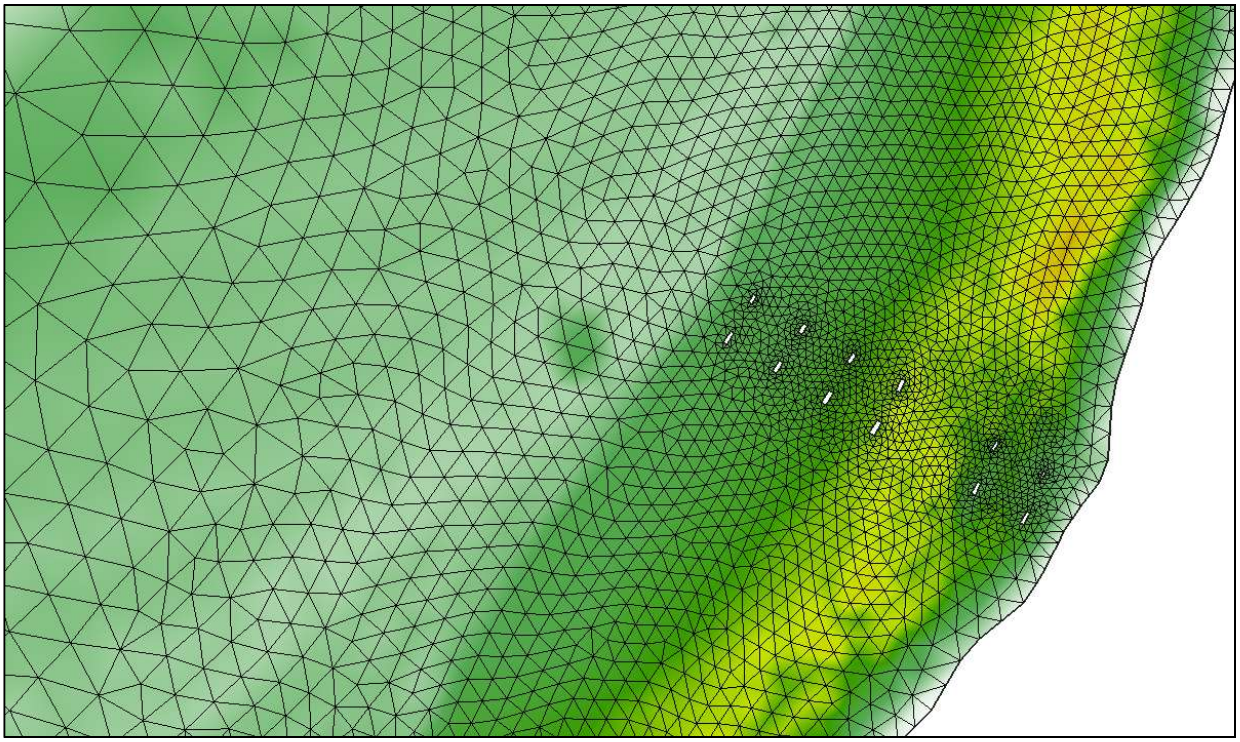

Figure 6-18. Example 2D mesh with a detailed mesh of increased resolution around structures. Green to red color indicates the terrain (low to high elevation).

Defining Roughness Coefficients

Defining friction values (roughness coefficients) spatially is very different between 1D and 2D modeling approaches. Both approaches can be based on estimating initial values from land use/cover data and aerial photography. However, this estimation approach is not required for 1D models. Instead, for 1D models, roughness coefficients are entered for each cross section, generally with a minimum of left overbank, channel, and right overbank values. Moreover, roughness coefficients can be further discretized horizontally for both the over bank areas and the main channel. Also a common approach for 1D models is to have options to vary roughness vertically at cross sections, or to be able to vary roughness with changes in flow rates.

Additionally, some 1D models even have options for changed roughness due to seasonal vegetation/temperature changes. While getting detailed initial roughness estimates into a 1D model can be somewhat time consuming, changing roughness for model calibration is very easy. With the use of tables to display and modify roughness values, larger portions of a model can be changed quickly, and a model can be calibrated in an iterative manner. Furthermore, flow vs roughness and seasonal roughness factors also add to the ease of calibrating a model, as well as testing the model's sensitivity to the calibrated roughness values.

Developing initial estimates of roughness coefficients for 2D models is similar to 1D, in that the modeler can use land use and aerial photography information to create spatial layers that can be related to initial estimates or roughness coefficients. Additionally, most 2D models have a way to override the initial roughness coefficients with user defined polygons. These polygons can be used to set better initial estimates of roughness coefficients, for example defining roughness for main channel areas, or the polygons can be used to change initial estimates during the calibration process.

Changing roughness coefficients for the calibration process is generally much more time consuming and difficult for 2D models than for 1D models. The reason for this difference is 2D models require roughness to be defined for every face of the 2D mesh. Furthermore, the calibration process is harder for 2D models because decisions must be made as to what spatial area extent require the roughness changed in order to calibrate a model.

Hydraulic Structures

Defining and modeling hydraulic structures can be very different between 1D and 2D modeling approaches. In general, 1D modeling is often based on semi empirical equations and coefficients to model hydraulic structures. The hydraulics of the structure may be completely defined with a single or family of rating curves. However, there are also applications where either the energy or momentum equation (using cross sections) is used to solve for the water surface and velocity up too, through, and out of the structure. For this type of modeling, the full momentum equation is required (not the diffusion wave form) in order to accurately capture the acceleration and deceleration through the structure.

For 2D modeling of hydraulic structures, a similar approach to 1D modeling can be taken, and most often is for the vast majority of structure modeling. However, there is the option to model the flow through the structure in an entirely two-dimensional approach, which is much more work and time consuming but yields much more detailed information. An example for applying this approach is for flow through a bridge. If the water is not going to reach the bridge deck and go pressurized, then a detailed 2D modeling approach can be used (using many small cells/elements). In this case, the purpose of the 2D modeling approach is to capture more detail of how the water surface and velocities change as the flow approaches the bridge, goes through the bridge, and expands as the water comes out of the bridge, as well as capturing flow separations and eddy zones.. An example of the detailed 2D modeling approach for a bridge is shown in Figure 6-19 (the figure shows the resulting velocities through a bridge from a detailed 2D model of the bridge).

This same approach can be used for many hydraulic structure types, depending on the desired level of detail. However, for structures where the flow passes through critical depth and goes into a free fall, acting like a waterfall, this approach will often produce incorrect results. The shallow water equations, as defined and utilized in almost all 2D programs are defined for gradually varied flow, and under the assumptions of a consistent hydrostatic pressure distribution through the flow field. These two assumptions are not true for flow passing through critical depth and going into free fall.

Even when the overall flow of a river is steady (inflow = outflow), the flow field in the vicinity of a hydraulic structure may be unsteady. This phenomenon can be captured with 2D and 3D models, but not with 1D models.

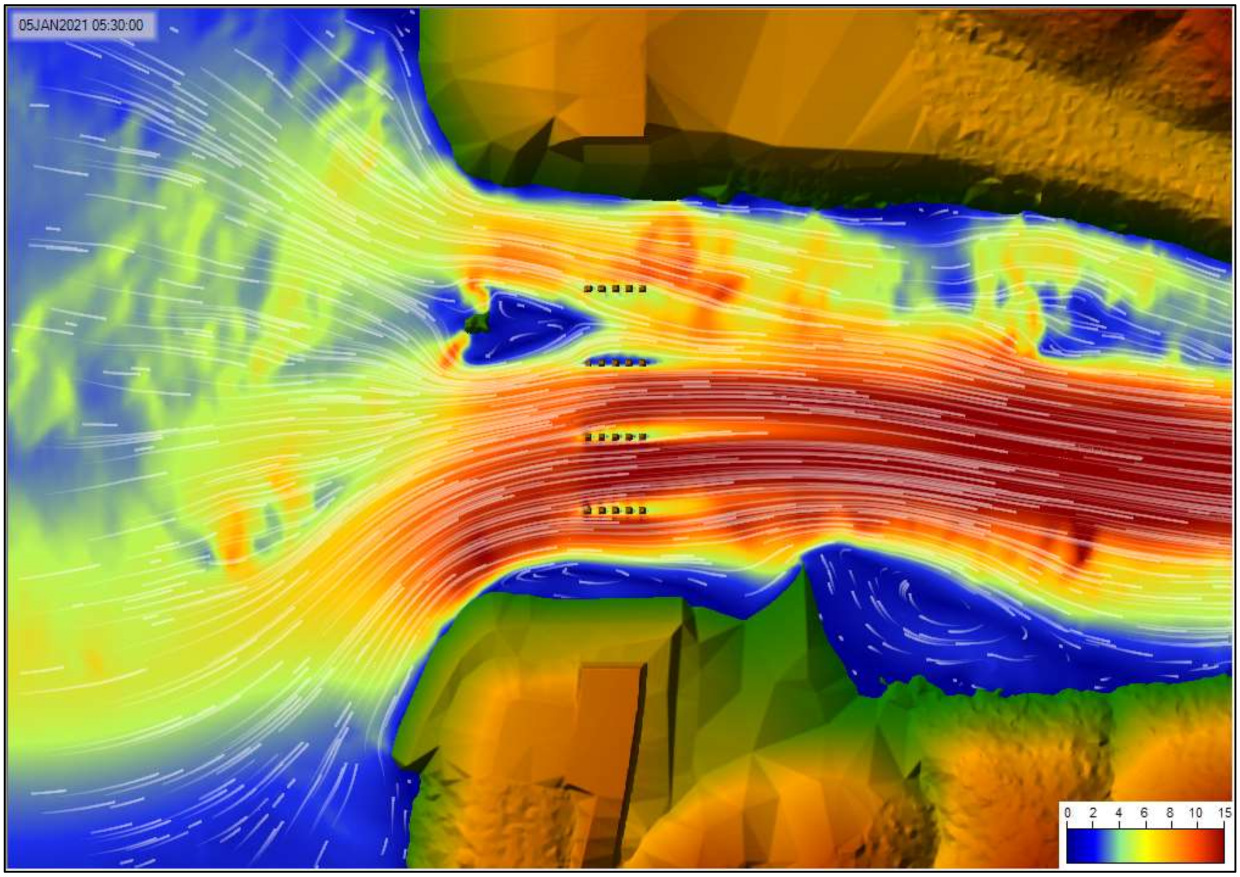

Figure 6-19. Example velocity output for a detailed 2D model of a bridge (velocity overlays terrain where green to red color indicates low to high elevation).

Boundary Conditions

Defining boundary conditions is similar between 1D and 2D modeling approaches. In 1D modeling approaches, all boundary conditions are either attached directly to a cross section, storage area, or an internal hydraulic structure. In 2D modeling approaches, boundary conditions are attached to outer boundary cells/elements, internal cells/elements/faces, and also internal hydraulic structures. All of the same types of boundary conditions can be used: flow, stage, normal depth (Manning's equation); rating curves; precipitation; groundwater; and ways to control hydraulic structures during the simulation. The only difference is that in 2D modeling, some of the boundary condition types allow for varying water surface elevations across the boundary condition location (i.e., normal depth outflow). Additionally, some 2D model boundary condition types may also require velocity information in order to capture the momentum of the flow more accurately as it comes into the system.

An additional consideration for choosing the location of the boundary condition in a 2D model, is the need to move that boundary condition far enough away from the area of interest to be confident that the location of the boundary condition is not influencing the results. An example might be a stream with a large overbank on one side where the upstream boundary condition is being applied. Applying the flow to that boundary condition that is being treated as a 1D element might bias the flow distribution in the overbank for some distance downstream of the boundary condition as the assumed 1D flow distribution is being propagated downstream within the 2D domain.

Model Calibration

The model calibration process is one of the most important steps in the development of a hydraulic model. Calibration of any hydraulic model is required in order to understand if the model is capable of reproducing past floods, and if it can predicting future flood events with confidence. The calibration process also allows for greater understanding of the models sensitivity to the data, friction forces, and other empirical coefficients. A model that is not calibrated is just a numerical experiment. Further, an uncalibrated model may or may not be even close to reproducing realistic water surface elevations and flows throughout the system. However, sometimes in emergency situations, where there is either no data to calibrate the model, or not enough time for model calibration, results from an uncalibrated model are better than no results at all.

The process of calibrating a model is very similar between the 1D and 2D modeling approaches, however, 2D model calibration can be more difficult and time consuming. The data required to perform a model calibration is mostly the same, except more detailed calibration of 2D models requires detailed velocity distribution measurements to ensure the accuracy of the velocity profiles which may be available at gaged locations where direct measurements are being made for a range of flows by agencies such as the US Geological Survey. The amount and quality of the data available will vary from location to location. In some instances there may be insufficient observed data to perform a full calibration, but instead may be better characterized as a limited calibration of certain aspects of the simulation or in limited parts of the model domain.

The general data for model calibration is: observed land cover/land use (possibly for different years or time of the year); rainfall records (or gridded rainfall data); observed water surface elevations at stream gages; computed flows at stream gages (based on observed water surface and an established rating curve); high water marks from debris and water stains; inundation extents/boundaries from aerial photography and field inspections; anecdotal accounts/water elevations from field investigations of homes and businesses that were impacted during historic flood events; and any photos or videos that were taken during the flood event, or shortly after. All of this information is used in either of the 1D or 2D model calibration process. In addition to this data, any detailed velocity measurements that are taken at cross sections, can often be very helpful for further calibration of a 2D model. This type of velocity information is usually gathered at structures of interest, or locations of interest, such as: bridges, culvert, weirs, spillways, sharp bends, around levees, and also near river training structures. This process often requires a critical look at all of the data being relied upon. Stream gages record stage, which is converted to a flow rate using a rating curve that may have bias. The water level measurements can be subject to mechanical problems or have localized hydraulic influences on the reading created by the structure they are attached to, etc. High water marks can be suspect if they are collected in areas where wave action, superelevation or other factors influence the water levels at that location and the elevations obtained with a hand-held GPS unit that has limited vertical and horizontal accuracy.

The process of calibrating a hydraulic model starts with the hydrology. A hydraulic model is only as good as the provided flow boundary conditions. Having good boundary conditions is imperative in any type of land surface and riverine modeling. Generally, a hydrologic model will be used to perform the modeling of the entire watershed, and provide flow data to the hydraulic model for boundary conditions. So the first step is to ensure that the hydrologic model is well calibrated. During the hydraulic model calibration, it may be necessary to go back to the hydrologic model for further calibrations/changes based on the results of the hydraulic model routing and computations of water surface elevations, flows, and hydrograph timings. This should be thought of as a common step, in which the hydrologic modeling and hydraulic modeling calibrations are done together, not as separate exercises.

Calibrating of 1D modeling reaches is accomplished by changing/adjusting the following:

- Hydraulic roughness parameters.
- Contraction and expansion coefficients.
- Ineffective flow area extents and height trigger elevations.
- Hydraulic structure coefficients.
- Bend loss coefficients (sometimes called minor losses).
- Boundary condition information, such as energy slopes, or even potentially rating curve values.
- Debris blockage information at structures.
- Levee breach dimensions and timing values.

Model adjustments of roughness coefficients are made on a reach basis, rather than individual cross sections. Changes should be made gradually between model runs, in order to get a better understanding of the effects of those changes on both the water surface elevation, but also hydrograph attenuation and travel time with the ultimate goal of making the model more closely mimic the physical setting and flow characteristics. Changes made upstream in a model will locally affect the water surface elevations, but will also affect the magnitude and timing of flows that go downstream. Changes made downstream in a model will not only affect water surface elevations locally, but may also affect the water surface elevations upstream for some distance. The flatter the stream, the greater the distance that will be affected by changes in water surface elevations downstream. An example of a calibrated 1D model river reach for the Lower Columbia River is shown in Figure 6-20. Displayed in Figure 6-20 are both observed data from gages, as well as high water marks collected after the flood. There is some noticeable scatter in some of the high water mark data, which is common, as this type of observed data can have considerable scatter and uncertainty in the values.

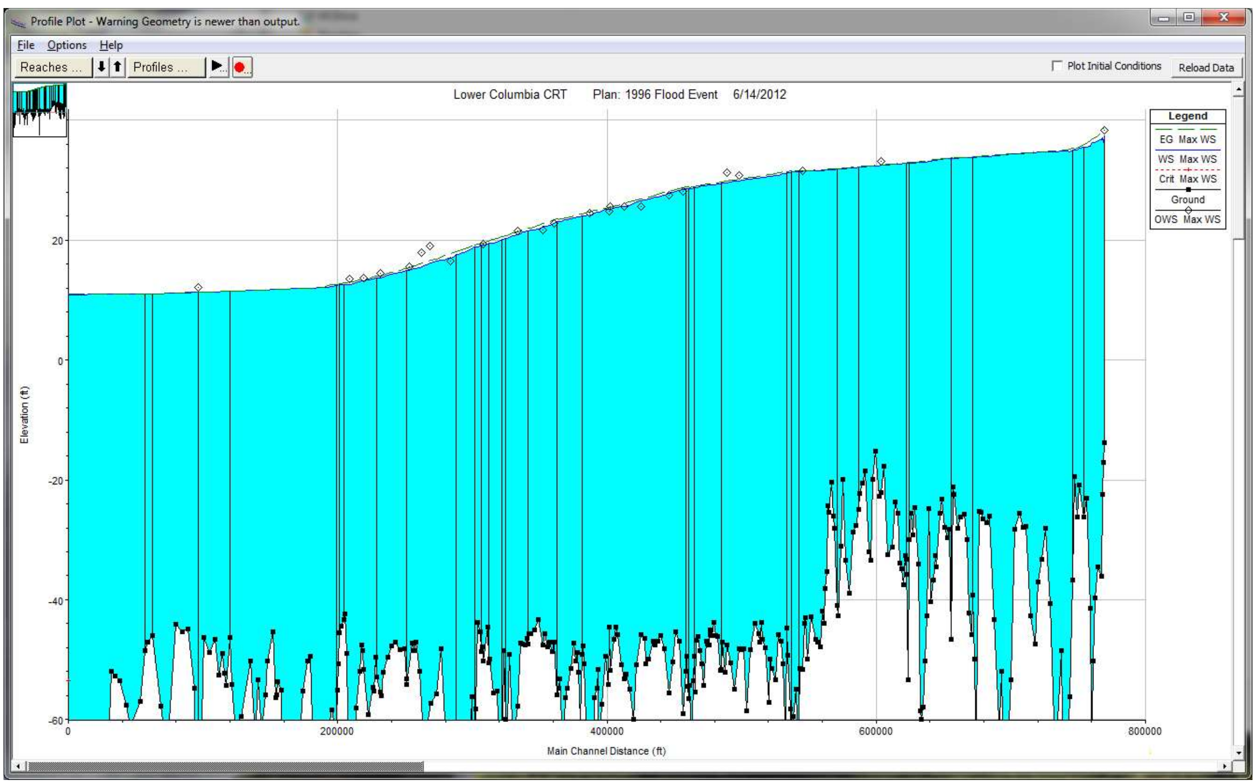

Figure 6-20. Example of a Calibrated 1D model for the Lower Columbia River.

Calibrating a 2D model is very similar, but with some of the following differences. In 1D modeling, changes to roughness are done at cross sections and/or groups of cross sections within a reach. In general, adjusting the roughness coefficients, either up or down as necessary, for a range of cross sections is easy by using tables of roughness values and factors to increase or decrease values by a percentage. Additionally, roughness factors that vary with flow and/or season are generally available and easy to use.

Adjusting roughness in a 2D model is more difficult, because the modeler not only has to pick a distance along the river to make adjustments, but also decide which portions of that area should be adjusted. This increased complexity is due to the much greater spatial variability that is defined for 2D modeling roughness coefficients (i.e., each cell/element face has potentially a unique roughness value). In addition to the spatial extent for making adjustments to roughness coefficients, another necessary decision is how much of an adjustment should be made to each of the base roughness land use types.

While 2D models do not use empirical contraction and expansion coefficients, and there is no need for ineffective flow areas and blocked obstructions at cross sections, there are empirical turbulence coefficients that may need to be adjusted, in order to better account for the extent of turbulent eddy zones, as well as natural diffusion of the flow field as the hydrograph travels downstream. How much of the diffusion of a hydrograph is due to frictional forces verses turbulence is debatable. Calibration of 2D models will also require the same types of changes as 1D models for the following aspects: changes to hydraulic structure coefficients that are affecting the results; boundary condition changes; debris blockages, and levee breaching information.

As mentioned previously, further refinement of 2D models can be made with observed velocity measurements. If a 2D model is being developed in order to gain detailed knowledge of the velocity distribution in specific areas, then having observed velocity information, even if the information is for lower magnitude events, can be very helpful in ensuring that the model produces reasonable results. Calibration to velocity information will generally require changes in roughness, possible eddy viscosity coefficients, and maybe even terrain adjustments, if appropriate and justified.

Time and Cost Issues

The time to develop a model with the 1D or 2D modeling approach can vary, depending on the type and purpose of the model. For example, if a "Quick and Dirty" model needs to be developed during a flood emergency, then it is much faster to lay out a 2D flow area polygon, set a basic cell/element size, attach some boundary conditions, and begin making simulations. Making a quick model with the 1D approach will definitely take more time. However, if a detailed model is being developed, then both approaches will ultimately take about the same amount of time to develop the initial model. Yes, there is a considerable amount of work required to lay out cross sections for a detailed hydraulics model. However, a lot of work is also required to refine a 2D flow area with additional mesh resolution. Further, some models might require the delineation of breaklines to define 2D flow areas. Estimating the base parameters, such as: roughness coefficients, hydraulic structure coefficients, and other empirical coefficients needed for the modeling approaches is about the same amount of time.

Furthermore, another factor that varies the model development time is the model calibration process. During this process adjustments are made to the model and then the model is computed for all of the desired events and scenarios. The main time discrepancy is due to the fact that 2D models take significantly more computation time then 1D models. For example, it is common for a 2D model to take several hours of computational time, possibly even a day or more. On the other hand, 1D models generally run in minutes to hours. Because of this significant computational time difference, the process of calibrating and performing the alternative analyses for a range of events, will be much more time consuming using the 2D modeling approach.

In addition to the computational time requirements, the hardware requirements for running 1D and 2D models is generally different. In general, 1D models are single threaded, meaning they require only a single core processor. On the other hand, 2D models can generally take advantage of as many core processors as there are available on the machine. For serious 2D modeling, the modeler should obtain a high level computational machine for such work, or even investigate HPC (High Performance Computing) and Cloud computing options.

Modeler Knowledge, Skills, and Abilities

Whether performing 1D or 2D modeling, the modeler should have a good background in hydrology, hydraulics, hydraulics of structures, and numerical solutions of non-linear equations. Additionally, no matter what computer model and modeling approach is selected, the modeler should take classes on how to use that specific piece of software effectively for 1D and/or 2D modeling. Self-study is great to get started on using a specific piece of software, but the additional knowledge gained from taking a class from the software developers/experts in using the software will often prove to be invaluable.

Gaining knowledge, skills, and abilities in modeling with either 1D or 2D modeling approaches is a matter of time and opportunity. The modeler needs to perform detailed studies of river systems in which they have good terrain, structure information, and historic data to perform model calibrations. If the modeler is new to modeling, they should always seek out guidance and assistance from experienced modelers. Additionally, all models should be reviewed by independent experts in order to ensure that the work performed was take with an appropriate modeling choice (1D or 2D); applied correctly; contains the appropriate amount of detail for the given location and study type; was calibrated effectively, and is reproducing reasonable hydraulic results for the events and alternatives being modeled.

Summary of 1D and 2D Modeling Advantages and Disadvantages

There are definite advantages and disadvantages to both the 1D and the 2D modeling approaches. This section of the document summarizes the advantages and disadvantages that have been described previously in this document. The discussion is based on advantages and disadvantages from the perspective of 1D modeling versus 2D modeling.

1D Modeling Advantages:

- In general, 1D models require less terrain data, in that the channel portion of the model can be from separate detailed cross sectional surveys.
- Often (but not always) 1D models are easier to calibrate, due to the simplicity of changing parameters such as roughness coefficients, and other variables.
- Modeling of hydraulic structures is often easier, requiring less data and computational requirements.
- Significantly less computational time and resources are required for 1D models.

- The 1D modeling approach may be more appropriate for areas in which the basic data does not support the ability to develop a reasonable 2D model.

1D Model Disadvantages:

- The flow path of the water, for all events, must be known before developing the model. The flow path is not always possible to know, especially in flat areas making those choices often subjective and less accurate.
- To get an accurate representation of the true flow area 1D cross sections must be laid out perpendicular to the flow. However, accomplishing this requirement is not always possible for the full range of events, and therefore may require more than one geometric representation of the system for low flow and high flow events which have different flow patterns.
- Energy and/or force losses due to contractions and expansion require the modeler to define empirical coefficients and ineffective flow areas.
- The direction of the flow during the event is limited to the defined flow path of the 1D model.
- Velocity output is limited to average values for the main channel and overbanks. While further discretization of velocity can be calculated from 1D output, it is limited to the assumption that the flow is perpendicular to the cross section, and the flow distribution is only a function of the cross section shape and roughness values.
- Mapping of the inundated area is based on the assumption that the water surface changes linearly between any two cross sections, and that the water surface is flat inside of storage areas.

2D Modeling Advantages:

- The flow path of the water, for all events, does not have to be known to develop the model. However, the extent of the flooding does need to be correctly defined.
- The direction of the flow can change during the event. Water can move in any direction, based on energy and momentum of the flow.
- Velocity, momentum, and the direction of the flow are more accurately accounted for with 2D modeling. This accountability is especially true for flow going over roads, levees, barriers, structures, around bends, and at flow junctions/splits. Additionally, 2D models can be used to analyze eddy zones within the flow field. Around bends, 2D models produce accurate water surface elevations, but velocity distributions might be erroneous due to the existence of helical flow.
- Energy and force losses due to contractions and expansions, etc. are directly accounted for, and do not require empirical coefficients, increased roughness, or user defined ineffective flow areas.
- The mapping of the inundated area, as well as velocities, and flood hazards (depth x velocity) is more accurate.

- Detailed modeling of hydraulic structures, in a full 2D modeling approach, can provide more insight into the flow distribution approaching, going through, and coming out of a structure.

2D Model Disadvantages:

- More accurate and detailed terrain models are required in order to develop an accurate 2D model. The terrain must include the details of the channels at all locations within the model as well as correctly capturing features such as roads, berms and levees. Overly filtered LiDAR data sets or data sets that have been processed at too large of a grid size may not properly resolve these key terrain features that influence flow behaviors and patterns.
- Defining and modifying roughness values requires more spatial definition, and can be more difficult and time consuming during the calibration process.
- Requires significantly more computational time and/or computational resources. May require the purchase of a very high level computer (many cores, fast CPU's, lots of RAM, and fast hard disk), or utilizing HPC and cloud computing solutions.
- May require using larger grid sizes than desirable for the problem, in order to reduce the run times to a manageable amount of time.
- May not really produce better results, if the data used to perform the modeling (terrain, channel data, and roughness) do not support the level required for accurate 2D modeling.

The decision between the choices of 1D or 2D models must maintain fidelity to the goals of the project being investigated, and the principle of "simplest, and technically sound path to achieve the goals." The lack of data for a 2D model, or the availability of a 1D model does not, in itself, justify the choice of a model unless supported by goals of the project. For example, if flood extents are of interest for a river that includes river bends or other 3D features, a 1D or a 2D model might be sufficient even though velocity distribution is incorrect; on the other hand if sediment behavior is of interest a 2D model modified for helical flow or a 3D model might be more appropriate. The engineer must avail oneself of the appropriate knowledge of the system to successfully attain goals of the project.

Chapter 7

Two-Dimensional vs Three-Dimensional Modeling

The question of 2D versus 3D hydraulic modeling is a very complex question. The primary reason for this complexity is that most modeling where 3D is appropriate includes vast areas where 2D behavior dominates. Unfortunately, the 3D areas impact the 2D areas and vice-versa. This chapter primarily addresses only with 3D-hydrostatic modeling, however, three application examples utilize 3D-Non Hydrostatic models. ***Henceforth 3D will imply 3D-hydrostatic, unless otherwise noted.***

Two and Three-Dimensional Equations

The physical laws which govern the flow of water are: (1) the principle of conservation of mass (continuity), and (2) the principle of conservation of momentum. These laws are expressed mathematically in the form of partial differential equations, which will hereafter be referred to as the continuity and momentum equations. In the derivation of both the two-dimensional and three-dimensional unsteady flow equations used in this chapter, there are several assumptions made about the flow:

1. The flow is considered to be of an incompressible fluid.
2. The pressure distribution is considered to be hydrostatic
3. The vertical acceleration of the water is considered to be negligible.
4. The bed slope is considered to be mild.
5. The effects of boundary friction can be taken into account with flow resistance laws derived for steady flows (e.g.., Manning's equation).
6. Bousinnesq approximation is valid.

2D equations were presented, previously, in Chapter 6 and therefore, will not be repeated here. 3D flow equations are presented below.

Three-Dimensional (3D) Equations

The three-dimensional continuity and momentum equation can be written in partial differential equation form, with respect to Pressure (P) , Velocity (u, v, w), and are commonly shown in hydraulic text books as follows:

Continuity Equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

Momentum Equations:

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}-fv=-\frac{1}{\rho_0}\frac{\partial P}{\partial x}+\frac{1}{\rho_0}\left(\nabla.\mathbf{T_x}\right)+S_x$$

$$\frac{\partial v}{\partial t}+\frac{\partial (vu)}{\partial x}+\frac{\partial (vv)}{\partial y}+\frac{\partial (vw)}{\partial z}+fu-=-\frac{1}{\rho_0}\frac{\partial P}{\partial y}+\frac{1}{\rho_0}\left(\nabla.\mathbf{T}_y\right)+S_y$$

$$P(x,y,z)=P_a(x,y)+\int_z^{\eta} g\rho(x,y,z)\,dz$$

where:

u	= velocity in the x direction
v	= velocity in the y direction
w	= velocity in the z direction
$\mathbf{T_x}$	= Turbulent stresses in the x direction
$\mathbf{T}_y$	= Turbulent stresses in the y direction
f	= Coriolis parameter
P	= Pressure
ρ_0	= Reference density
ρ	= Spatially varying density
P_a	= Spatially varying atmospheric pressure
g	= gravity
S_x	= Additional momentum sources/sinks in the x direction
S_y	= Additional momentum sources/sinks in the y direction

The spatially varying density, ρ, is calculated using equations of state for temperature (T), salinity (S), sediment (Se) either individually or as a combination. The equation of state is represented as:

$$\rho(S,T)=\rho_0+\left[\beta_S(S-S_0)-\alpha_T(T-T_0)\right]+\delta\rho$$

$$\delta\rho=\sum_{i=1}^{i=n}\frac{C_i\left(SG_i-1\right)}{SG_i-C_i\left(SG_i-1\right)}$$

where:

β_S	= 0.78 kg m^{-3}/ppt
α_T	= 0.15 kg m^{-3}/ºC
$\delta\rho$	= density difference due to sediment
i	= sediment class, 1 to n
SG	= Specific Gravity
C	= Concentration of the sediment class

Figure 7-1 provides a diagram defining the symbols used in the 3D equations.

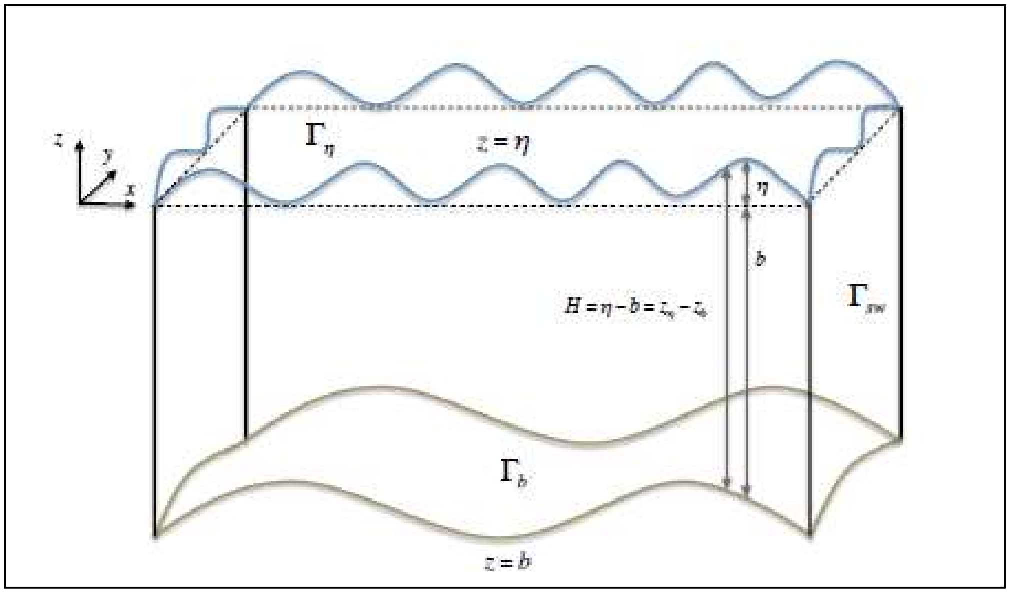

Figure 7-1. Definition of symbols used in 3D equations.

Application Examples

There is a distinct set of engineering problems/application that gravitate towards the application of a 3D numerical model – generally when the modeler is interested in a three-dimensional description of velocity (complex bathymetry, bends, or near hydraulic structures), density-gradient driven flows, or temperature-gradient driven flows. Below are some specific examples of these problems/applications.

- When modeling vertical stratification due to variable density (freshwater/saltwater): This is a common problem encountered in estuarine environments where the estuary is deep, or contains a deep navigation channel through shallow areas (e.g., Mobile Bay, Galveston Bay), Figure 7-2. In such applications, the saline water pushed in by the tides into the estuary causes the stratification of the estuary (Figure 7-3), with freshwater from terrestrial sources on the surface and saline water below. This stratification also results in the direction of flow between the surface and the bottom to be opposite at certain tidal conditions. Figure 7-3 also illustrates the variation in velocity between the surface and bottom. This variation in velocity is a condition that 2D models cannot simulate, 2D laterally averaged models can replicate this stratification but not the flow distribution between the deep channel and the shallows.

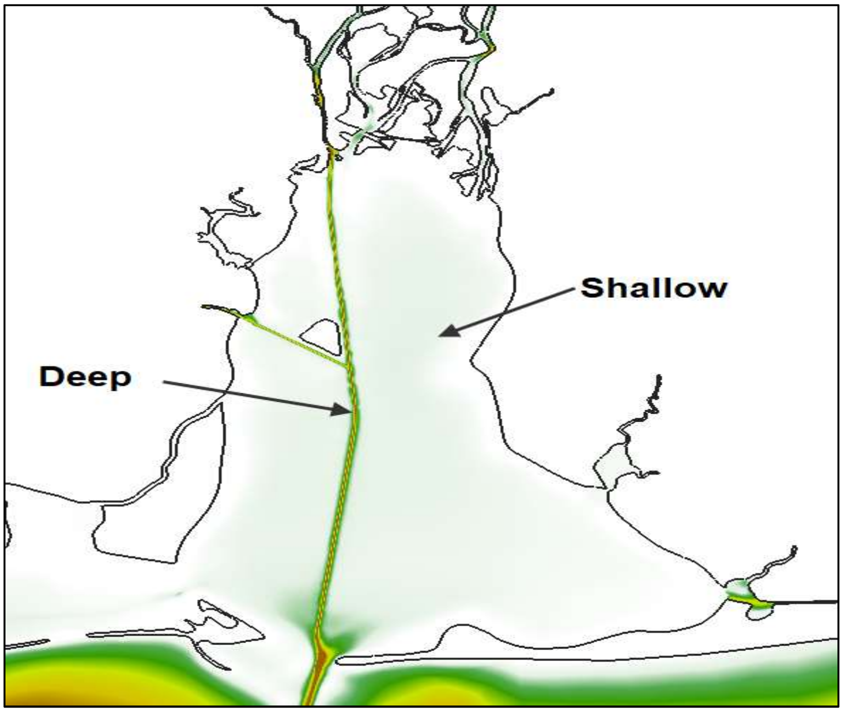

Figure 7-2. Deep channel surrounded by shallow areas, red indicates deeper and green shallower.

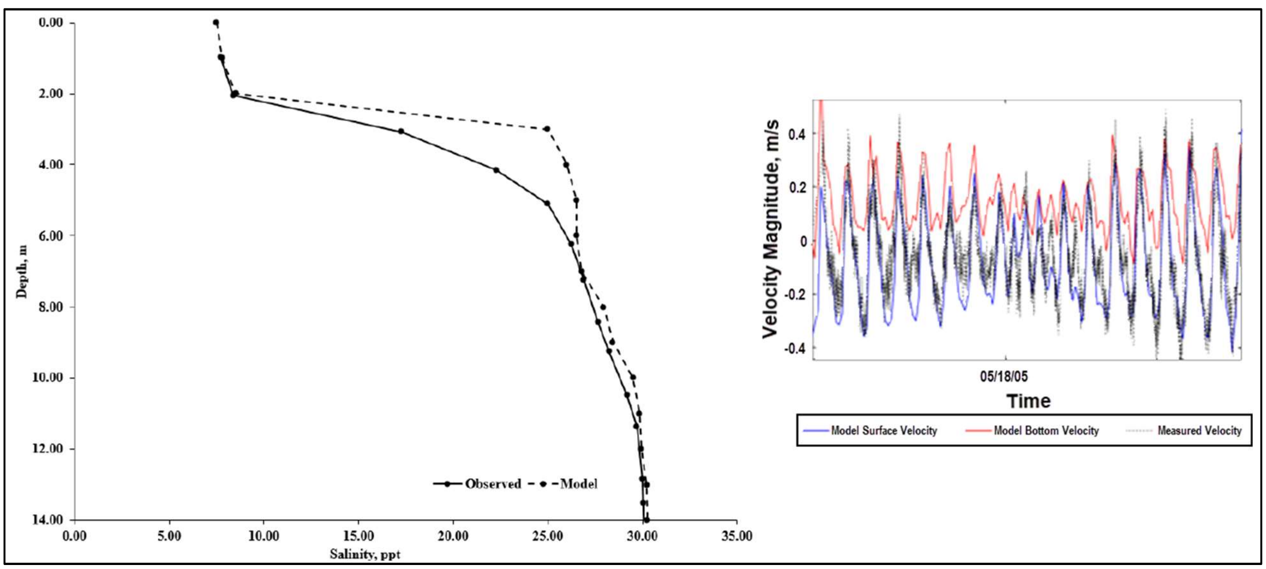

Figure 7-3. Salinity stratification and velocity differences in an estuary.

- When modeling stratification due to temperature: In a manner similar to salinity, temperature causes density differences between the surface and bottom. Temperature driven density differences is most often observed in reservoirs, and or lakes. Water is particularly receptive to this phenomenon, where during the hot summer the top of the water column is warmer than the bottom, and during the winter the surface is colder than the bottom.

The second scenario in particular is problematic, as it causes unstable stratification and eventually results in "overturning" of the water where the colder water sinks down and the warmer water rises up. "Overturning" has severe, and negative consequences for water quality and ecology. Furthermore, 2D-laterally averaged numerical models can simulate this temperature stratification of the reservoirs, but are inherently incapable of capturing the hydrodynamics associated with "overturning". Figure 7-4 provides an example of temperature stratification.

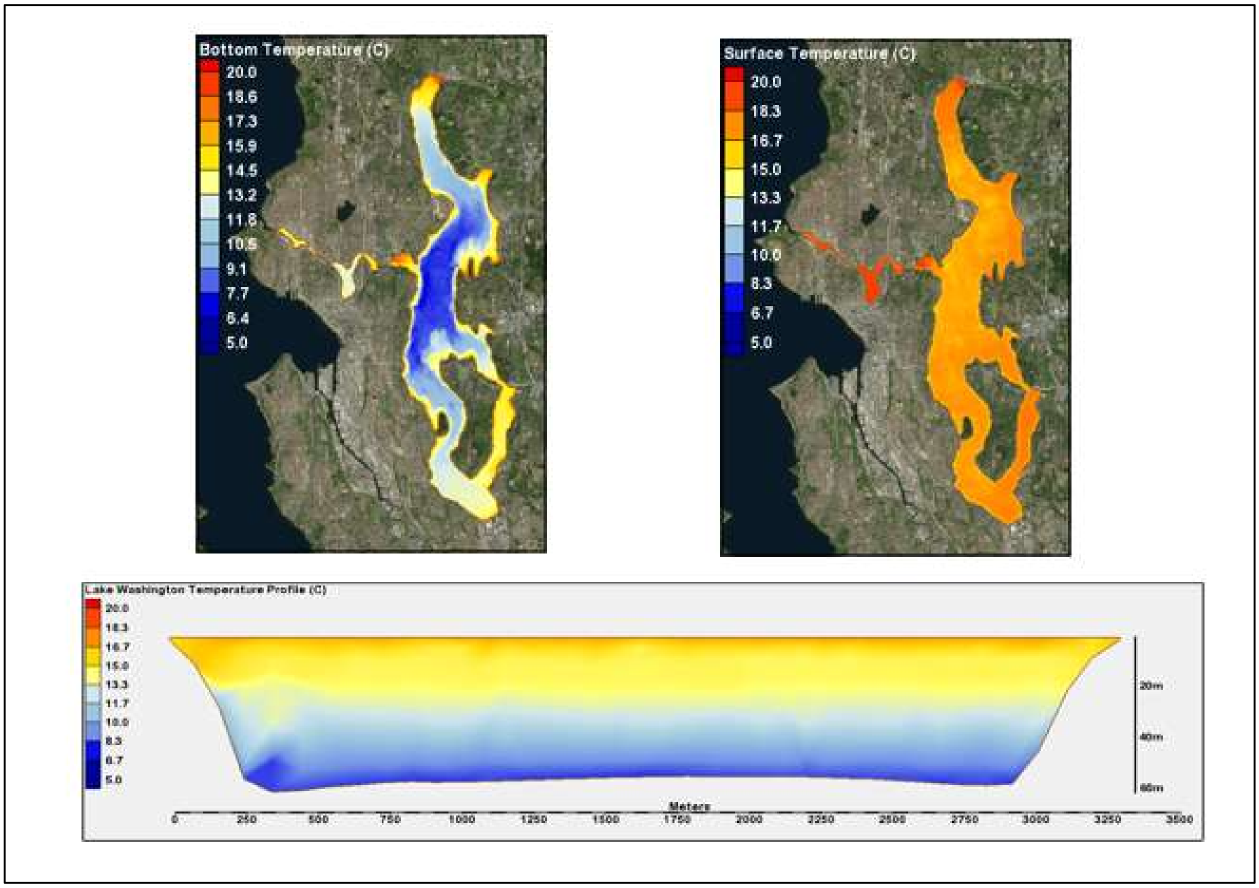

Figure 7-4. Temperature stratification in a lake.

- When modeling flow around a bend: The motion of an incompressible fluid, such as water, around a bend is governed by a balance between inertial and pressure forces. Inertial forces cause the water to flow towards the outside of the bend, and results in the creation of an elevation gradient. The outside of the bend attains an elevation higher than the inside of the bend. This "super-elevation" causes a pressure differential between the outside and the inside, resulting in the flow at the bottom being towards the inside and the flow at the top being towards the outside. This flow condition is called "helical." The generation of "super-elevation" can be simulated by 2D models, and the broad behavior of the momentum distribution due to helical flow can be simulated by the addition of momentum distribution correction terms to 2D averaged models.

Figure 7-5 presents a 2D depth-averaged velocity field corrected using the vorticity transport method (Bernard and Schneider, 1992; Finnie et al., 1999). The magnitude of the corrections applied to velocity field computed solely with the 2D shallow water equations is presented in Figure 7-6. Vertical variations in velocity may be estimated from these corrections for use in quasi-3D transport models (Brown. 2012). However, the determination of the exact distribution of velocity or proper behavior of sediment transport over depth as well as across the bend requires a 3D numerical model. Figure 7-7 and Figure 7-8 provide an example of this helical flow distribution.

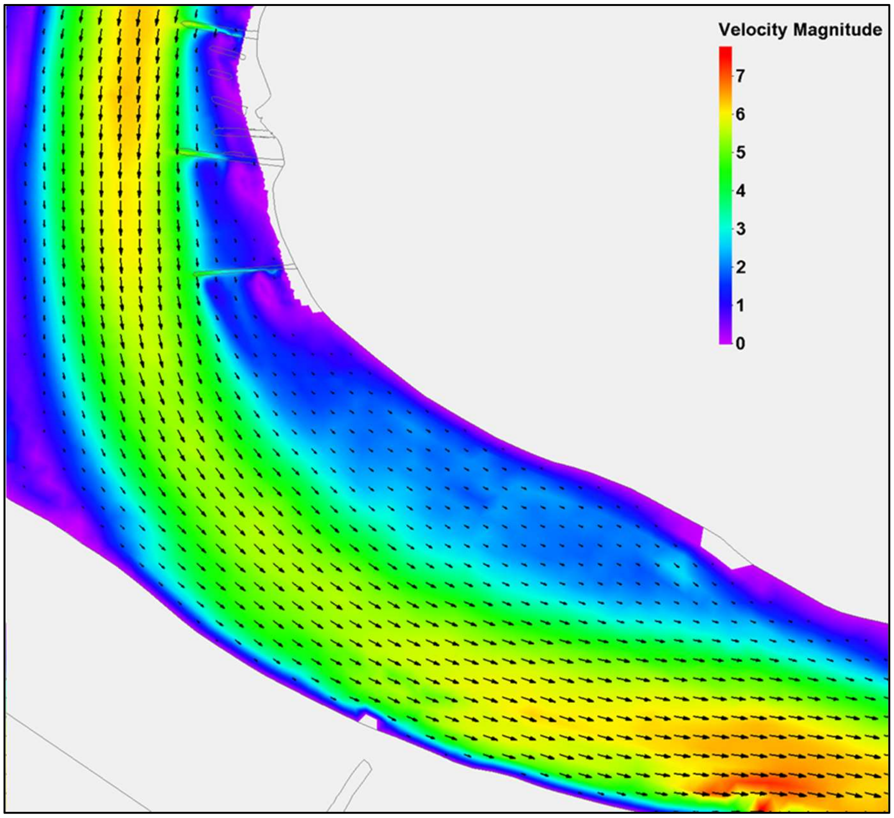

Figure 7-5. Computed depth-averaged velocity field in the Mississippi River with vorticity transport.

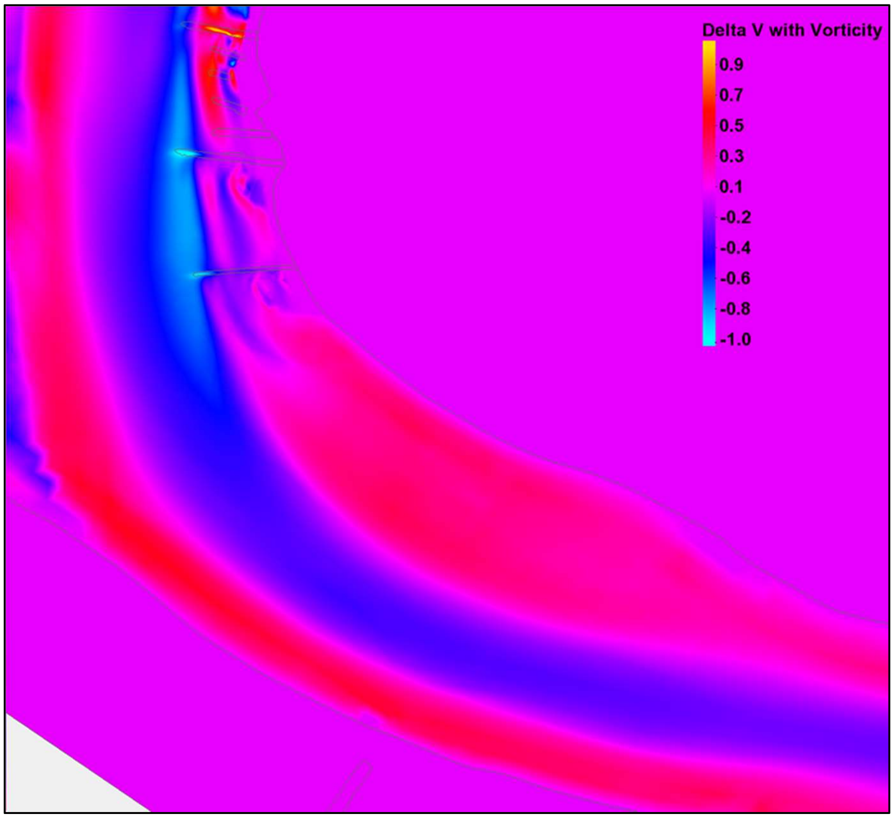

Figure 7-6. Change in computed velocity produced by vorticity transport method.

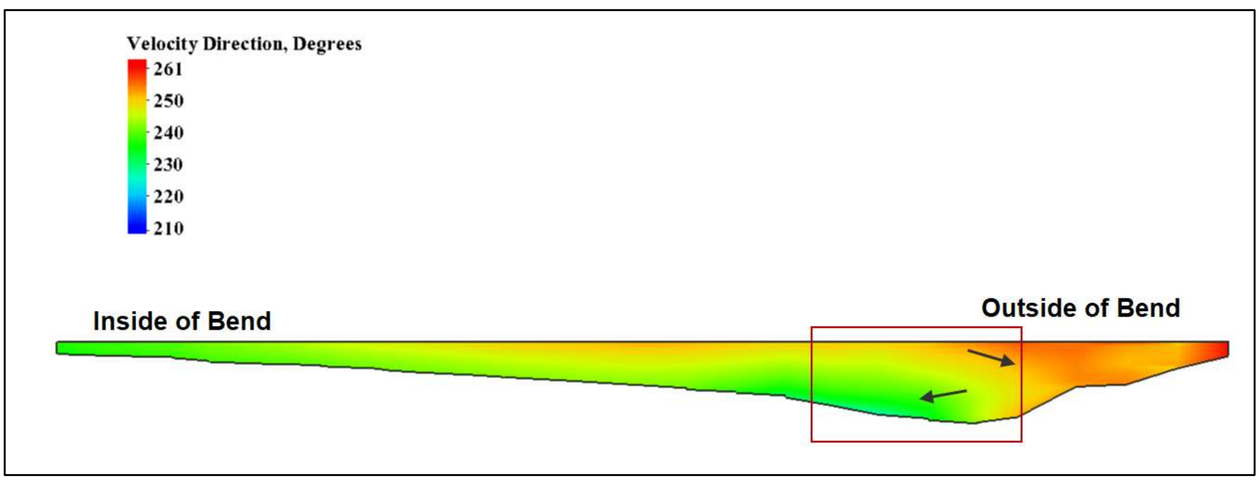

Figure 7-7. Helical flow in a river bend.

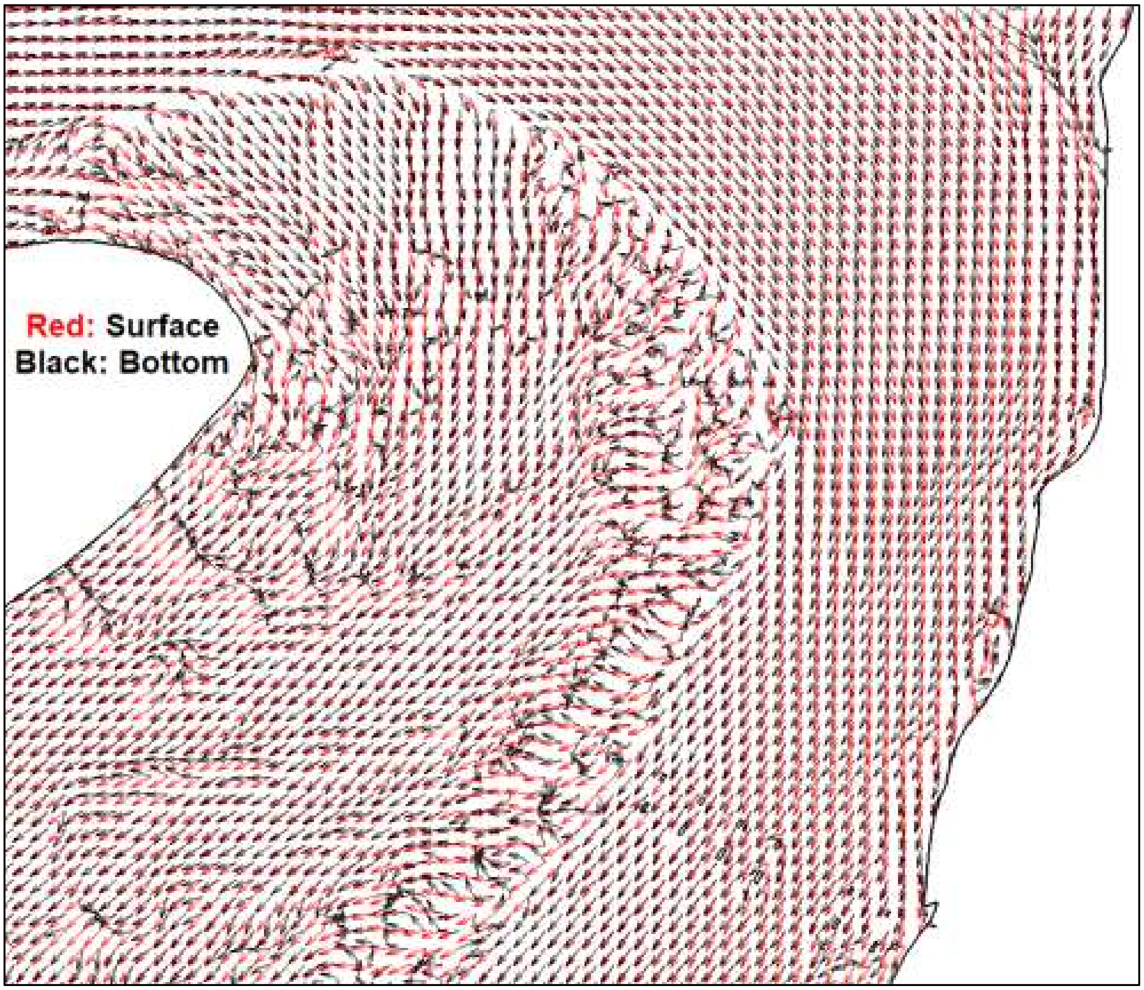

Figure 7-8. Velocity difference between surface and bottom in a river bend.

- Modeling Selective Withdrawal: Selective withdrawal strategies, for reservoir flushing, river diversions etc., are utilized to optimize management strategies. Selective withdrawal involves the removal of water at certain locations in the water column. The processes involved in this withdrawal are inherently 3D in nature and cannot be replicated or simulated using 2D or possibly 2D laterally averaged models. However, 2D models can be used to reduce the number of alternative designs to be simulated using 3D models.

One of the questions frequently asked by hydraulic engineers, planners and designers is "*can a 2D model be used to design gates, and/or spillway structures?*" The answer to this question is project specific, complex and experience dependent. For example, 2D models should not be used to design pressurized gates and/or spillways; however, 2D models are strongly recommended for reducing the design alternatives for which physical and/or 3D modeling is required. In general, 3D-Non Hydrostatic (3D-NH), and 3D-Non Hydrostatic Multi Phase (3D-NHMP) are recommended for these structures. Physical modeling of these structures is recommended in circumstances where the structure is exceptionally unique and no knowledge exists to gauge the performance of 3D-NH/3D-NHMP models.

3D Modeling of Structures

Three examples are presented below to illustrate the comparison of 2D and 3D models for such design efforts. **In the following three examples, the terminology 3D indicates 3D-NH and 3D-NHMP. The examples below assume that the 3D model results are accurate.**

- Modeling Flow through Gates: Modeling flow through gates for pool regulation, downstream flow management, etc. is a relatively routine part of water management and hydraulics. However modeling the relationship between flow quantity and upstream stage is not. This specific type of problem is handled through 3D-NHMP modeling, and physical modeling when the project complexities require. A commonly asked question is "*can 2D modeling be used to adequately guide selection criteria, or to reduce the number of designs to simulate using 3D?"* Consider a straight reach of a river with 9 gates wide open, which are required to convey flow under certain conditions. For comparison, Figure 7-9 provides the modeling results for this example by displaying the velocity results from 2D (depth averaged), and 3D-NHMP (unpressurized) at a vertical plane through the gate centers. The 2D velocity results insufficiently mimic the 3D-NHMP results, and the flow distribution through the gates and however the recirculation zones downstream of the gates is adequately reproduced (Figure 7-9).

  Now consider the same example except that the central gate intrudes 0.5 meters into the water column. Figure 7-10 illustrates the comparison of velocities between the 2D results, and 3D-NHMP pressurized simulations for this new situation where the central gate intrudes 0.5 meters into the water column. Again, the comparison indicates that the depth averaged velocity from the 2D model insufficiently mimics the velocities from the 3D-NHMP model (Figure 7-10). There are obvious difference in the pressurized zone, however results away from this area are not in concurrence with the 3D-NHMP simulation either. This modeling exercise, of a simple gate layout, indicates that "*yes, 2D simulations can be used to guide designs that should be simulated using 3D-NHMP, and designs that should be discarded from consideration."* However, 2D model results should "**NOT"** be used for final designs of gate openings.

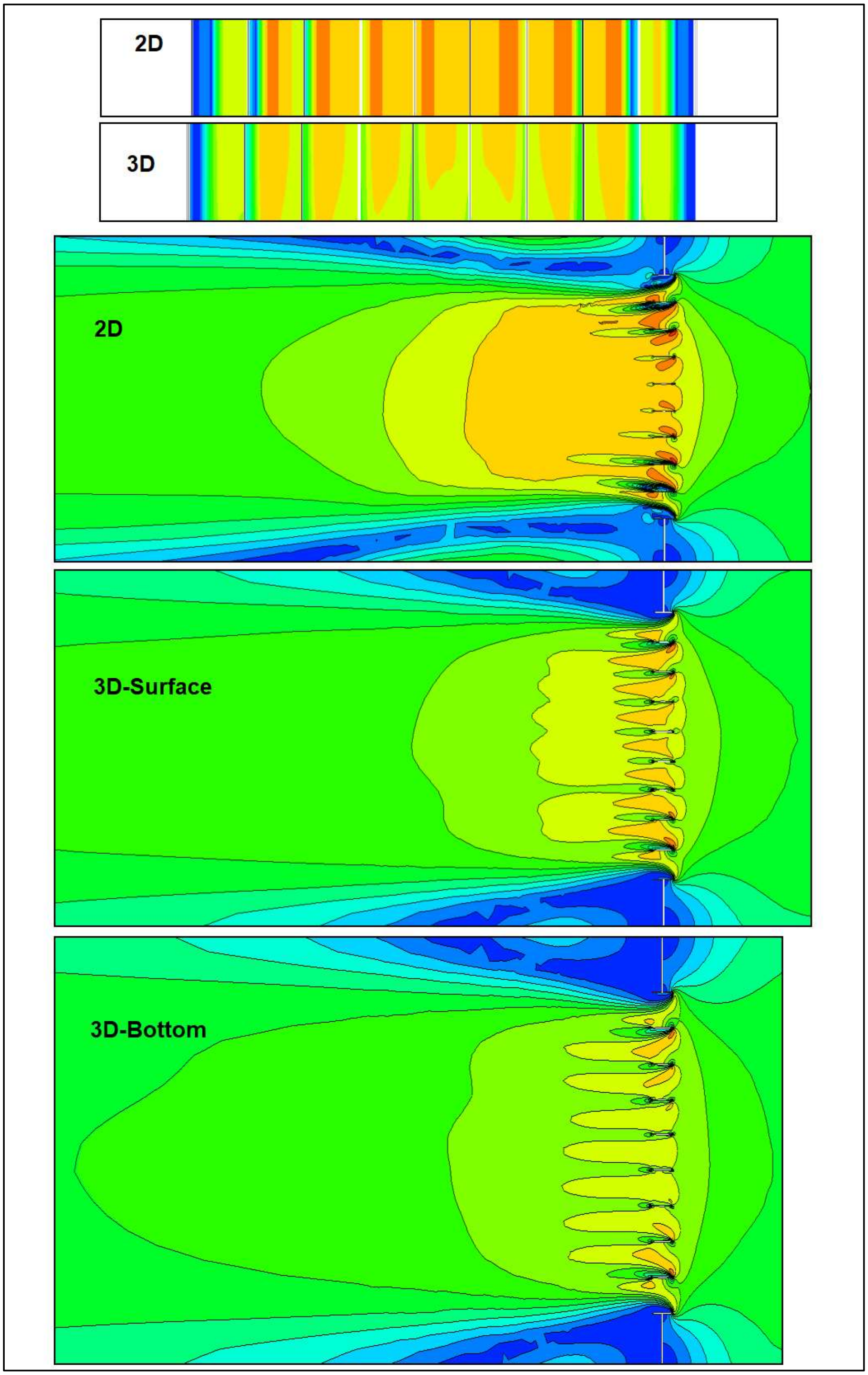

Figure 7-9. Velocity comparison between 2D (depth averaged) and, 3D-NHMP (unpressurized) models for a straight reach of a river with nine equally sized gates. Colors scaled to illustrate patterns, not exact values.

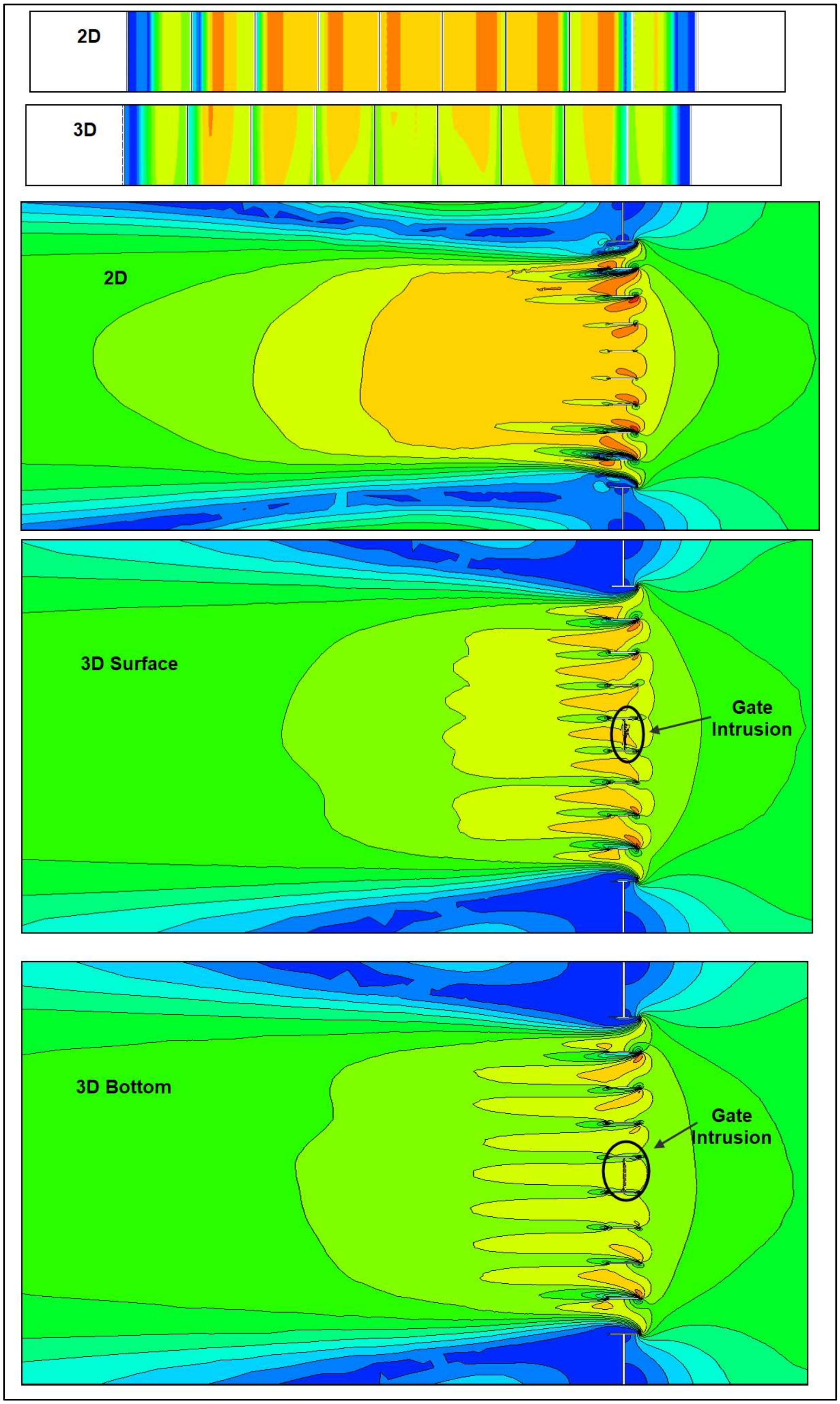

Figure 7-10. Depth averaged velocity comparison between 2D, and 3D-NHMP (pressurized) models for a straight reach of a river with nine gates where the central gate intrudes 0.5 meters into the water column. Colors scaled to illustrate patterns, not exact values.

- Design of simple sloped spillway: Spillways are essential reservoir management structures, and convey water to the downstream of the reservoir. An essential component of spillway design is the determination of hydraulic loadings, and the length of hardening required to prevent scour of the structure. The industry standard best practices for the design of spillways rely on the utilization of physical models, and/or 3D-NHMP models. The availability of 3D-NHMP numerical software has led to engineers utilizing physical models to validate the numerical model and then utilizing the numerical models to iterate on the optimum design. 3D models have a faster turnaround time, as well as lower costs than physical models but are several times more computationally taxing than 2D models. Therefore, there remains a temptation to utilize 2D models to further reduce time spent in design by using 2D models to eliminate obviously ineffective designs.

  Figure 7-11 shows the layout of a reservoir along with the location of the spillway and the outflow channel. The spillway must convey a Probable Maximum Flood (PMF) of 6,179 m^3/sec, and the design tailwater is 158 meters. Figure 7-12 shows the velocity results from a 3D model, and Figure 7-11 and Figure 7-13 show the results from a 2D model. Both the 3D and the 2D model show a similar velocity behavior. The 3D and the 2D model both simulate similar locations for the hydraulic jump, however the 2D model does not dissipate the same energy when compared to the 3D model.

  Further, the 3D model simulates velocities between 1-5 m/sec after the jump (Figure 7-12), whereas the 2D model simulates between 8-10 m/sec (Figure 7-14). This indicates, that if the 2D model is utilized to design the spillway, a longer hardened apron would be required, when as shown by the 3D results the energy would have been dissipated within a significantly shorter apron. This result is expected due to the fact that 2D models do not have all the mechanisms through which energy is being dissipated, namely the vertical mixing and the interaction between air/water/structure. Therefore, depending upon the flow regime, downstream control, necessary design parameters, and other characteristics the 2D model might be capable of sufficiently replicating the physics of the system, but not likely. Best practices indicate that the 2D model must be validated to a 3D model, and only then used to guide selection of designs that should be simulated using 3D.

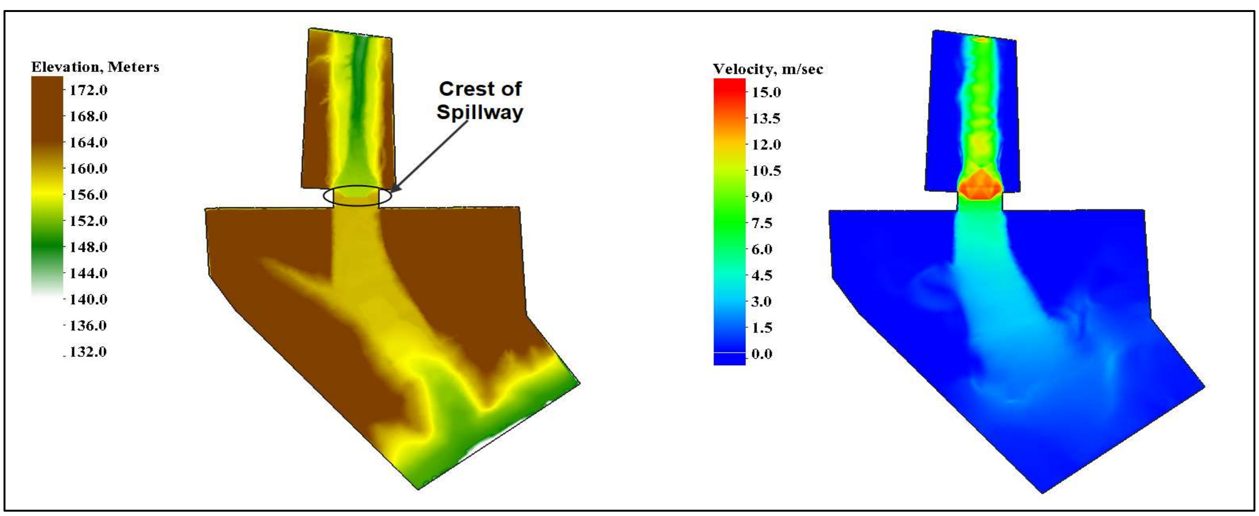

Figure 7-11. Bathymetry and 2D-velocity for a sloped spillway that must convey a Probable Maximum Flood (PMF) of 6,179 m^3/sec.

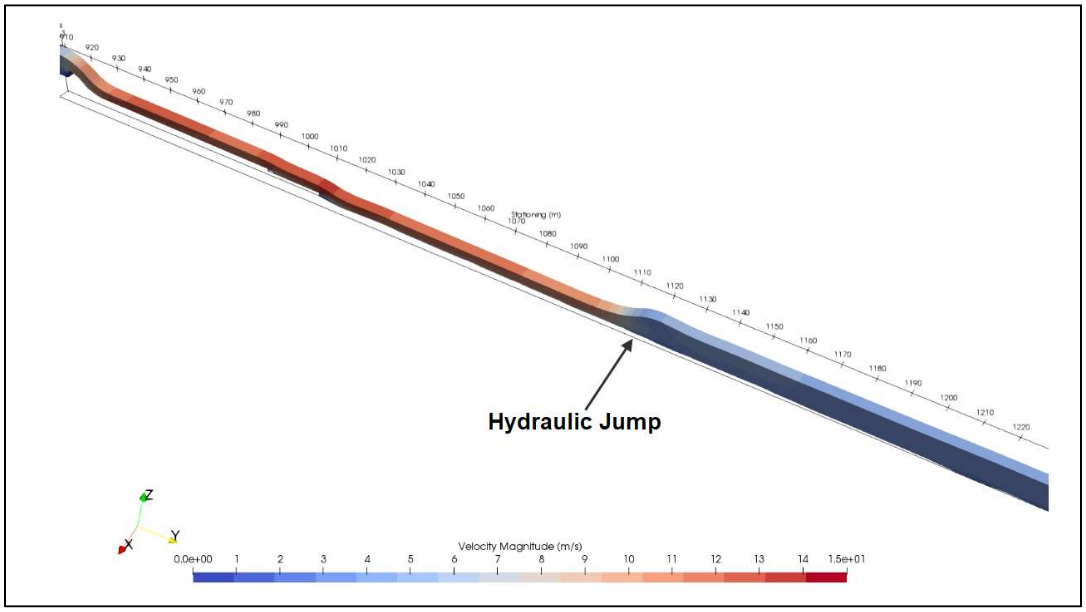

Figure 7-12. Sloped spillway (which must convey a PMF of 6,179 m^3/sec), 3D velocity (colors represent velocity) and water surface (thickness represents water surface) results.

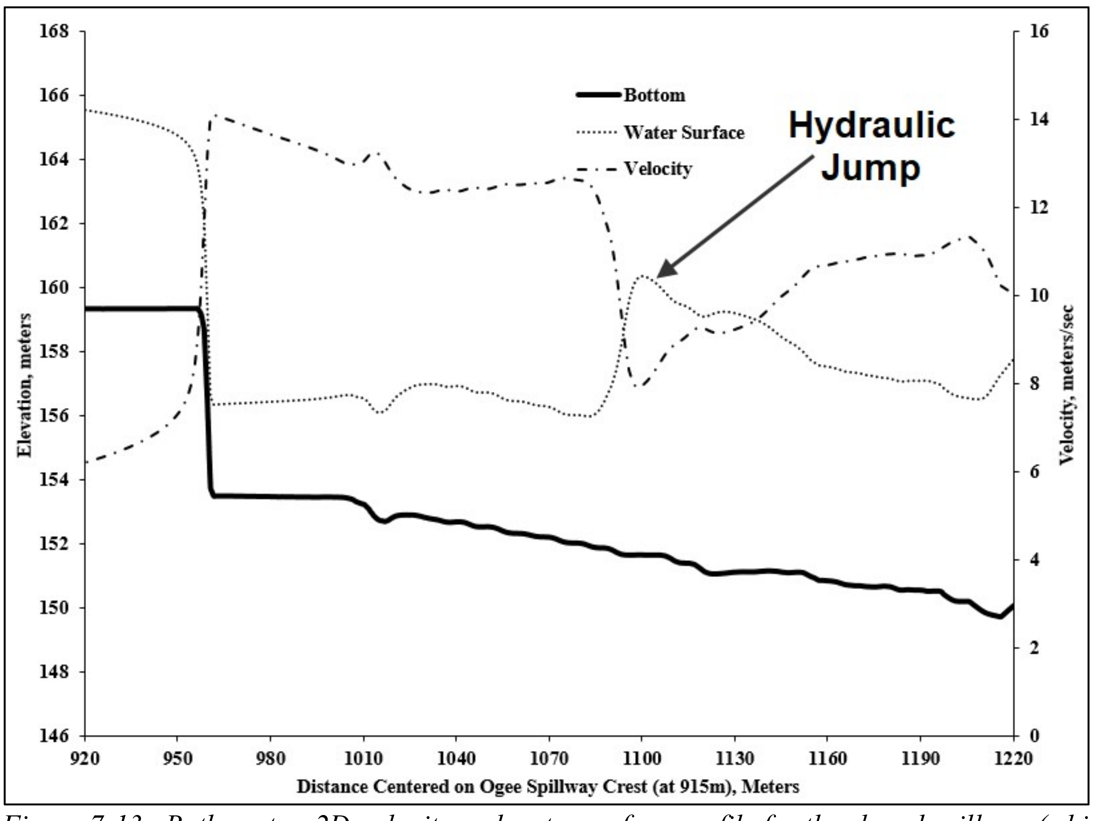

Figure 7-13. Bathymetry, 2D-velocity and water surface profile for the sloped spillway (which must convey a PMF of 6,179 m^3/sec).

- Design of stepped sloped spillway: Another common design of a spillway involves creating steps for energy dissipation prior to the stilling basin. Figure 7-14 shows an example of a stepped spillway. This example spillway, constructed of concrete, must pass a Probable Maximum Flood of ~3,145 m^3/s, and the tailwater control is at ~437 meters. Figure 7-15 shows the 3D-NHMP model simulated hydraulic behavior of the spillway, and the stilling basin, and Figure 7-14 shows the same for a 2D model.

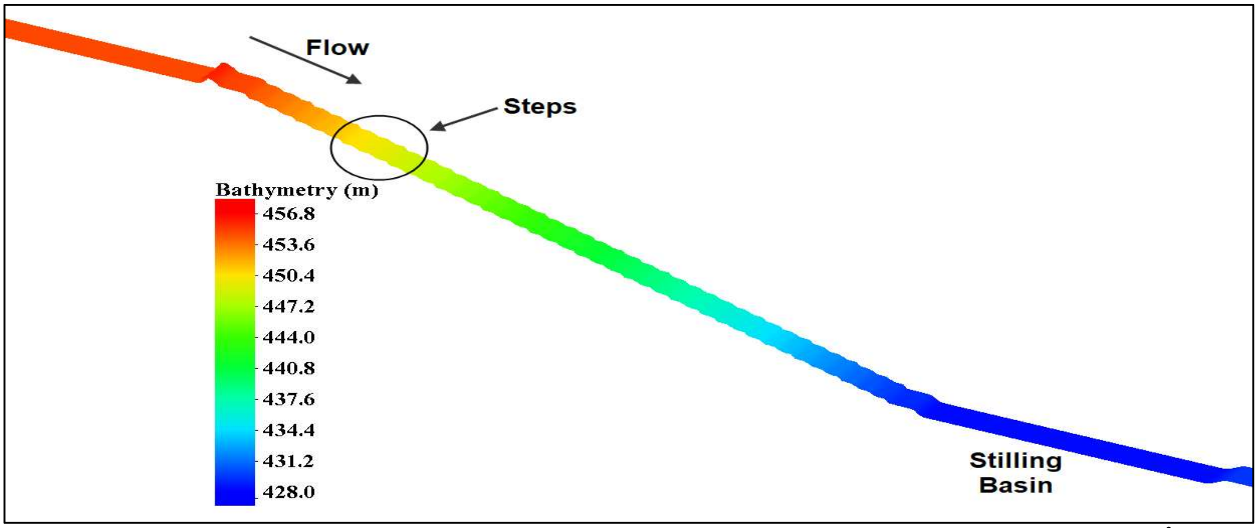

Figure 7-14. Bathymetry for a stepped spillway which must pass a PMF of approximately 3,145 m^3/s.

A cursory glance indicates that both models are providing similar hydraulics but on closer inspection observe that the maximum velocity simulated by the 3D-NHMP model is 10 m/s (Figure 7-15), compared to ~15 m/s computed by the 2D model (Figure 7-16). This difference in the maximum velocity is expected, for the 3D-NHMP model incurs energy loss due to water falling downslope over the steps, but the 2D model cannot account for these losses.

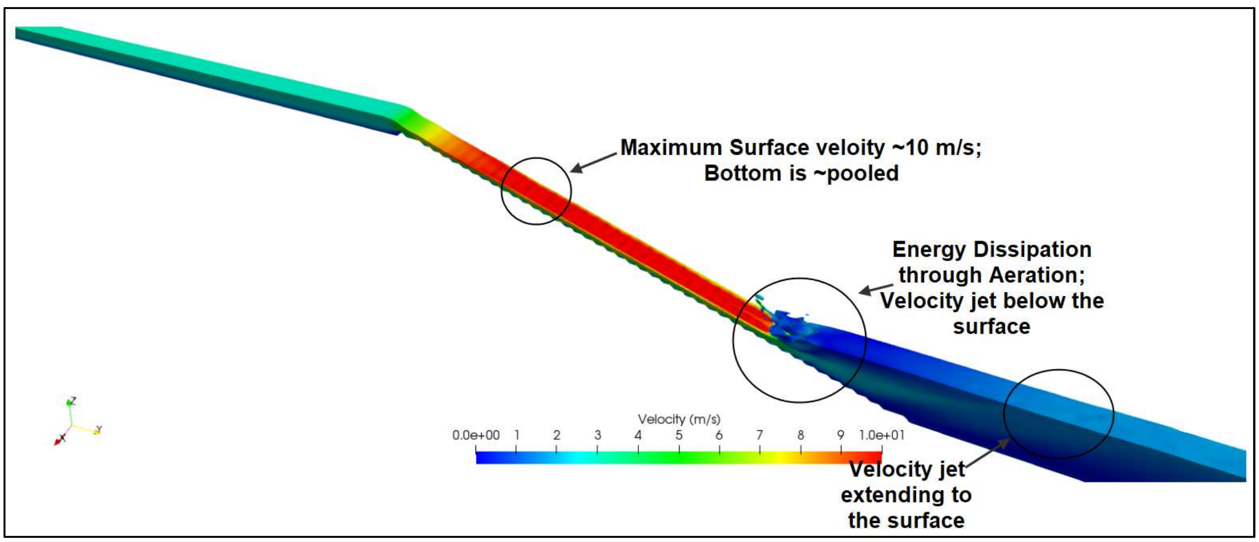

Figure 7-15. Results for the 3D-NHMP simulated hydraulics for a stepped spillway (which must pass a PMF of approximately 3,145 m^3/s).

It should be noted that if the steps are less severe the 2D results might provide similar results to the 3D-NHMP. The 3D-NHMP simulation shows that an undersurface jet of water exists at tailwater control, and the jet extends into the surface downstream, the 2D model is incapable of simulating this behavior. The 2D model shows the existence of a jump (Figure 7-16), which is not shown in the 3D-NHMP simulation. At this location the 3D-NHMP model shows the existence of air-water interaction, and air entrainment, the 2D model is incapable of simulating this behavior.

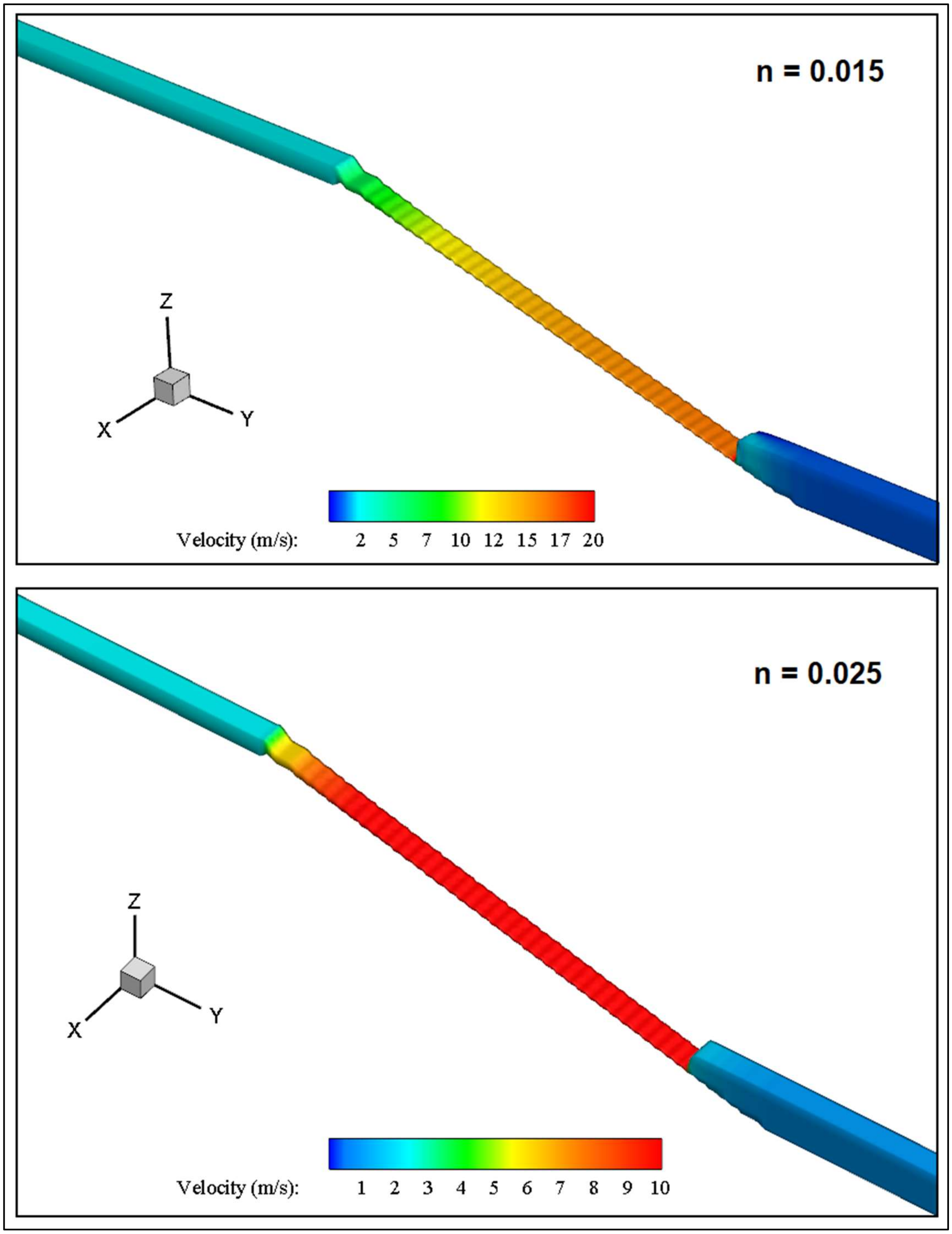

Figure 7-16. Results for the 2D simulated hydraulics for a stepped spillway (which must pass a PMF of approximately 3,145 m^3/s) for different Manning's roughness (n). Thickness indicates depth.

A good numerical modeler will know that the 2D model is incapable of accounting for all the losses in this system, and a modification of the roughness values might be undertaken. This decision is acceptable and justified within reason, and understanding that 2D modeling is being undertaken as a means to reduce the amount of physical and/or 3D-NHMP modeling, and not for final design. Figure 7-16, lower pane, shows the results with the Manning's roughness increased to 0.025 from 0.015. Note, that the velocities matches those reported by the 3D-NHMP model, the behavior at the toe is erroneous, but the average behavior on the spillway, not in the stilling basin, is correct. It must be noted that this increase in Manning's roughness is not physically justified, and should not be encouraged.

The three examples presented above illustrate that the 2D models are simulating behavior broadly inconsistent with 3D-NHMP models. A good numerical modeler will recognize that the 2D model is incapable of, implicitly, accounting for all losses in this system. In this case it is acceptable to increase model roughness, within reason, to obtain a better understanding of the system. Therefore, 3D-NHMP models are recommended for such design, supplemented with physical modeling if required. However, it is recommended that 2D models can be used as a preliminary design elimination tool to potentially reduce the number of physical models and/or 3D-NHMP simulations. However, 2D models should not be used for final designs of these types of structures (as shown above).

In the sections below 3D indicates 3D-Hydrostatic unless otherwise specified.

Model Development

While the development of a hydraulic model is similar, regardless of whether the model is 2D or 3D, there are also many significant differences in the model development process. There are also significant differences in model development between 3D hydrostatic and 3D non hydrostatic. This section discusses the model development process, and highlights differences between the two approaches – 2D and 3D hydrostatic. Additionally, advantages and disadvantages of the 2D and 3D modeling approaches are discussed.

Terrain/Bathymetric Data

As discussed previously, all hydraulic and hydrodynamic model development requires terrain data. 3D models require that a complete terrain or bathymetric model is built that includes data for the entire modeling domain. Depending upon the 3D mesh transformation strategy used, modification including smoothing of the bathymetry might be required. The section below provides examples of various mesh transforms used in 3D modeling. For details on terrain needs for 2D modeling approaches, see Chapter 4.

Mesh Transform

3D numerical models require a suitable method for tracking the transient free surface. Historically, a variety of coordinate descriptions has been used, the most popular being:

- Sigma (σ) transform (Figure 7-17A),

- Z-plane transform (Figure 7-17B) and,
- Arbitrary-Lagrangian Eulerian (ALE) (Figure 7-17C).

Each of these have advantages and disadvantages. Z-transform gridding is ideal 3D simulations where a denser fluid must remain trapped within the deep channel, σ-transform gridding is ideal for simulations where a denser fluid must fall down into a deep channel, and ALE gridding is conducive for both of the mentioned conditions. ALE gridding though advantageous for most modeling conditions, in general, requires higher resolution than both z- and σ–transform gridding.

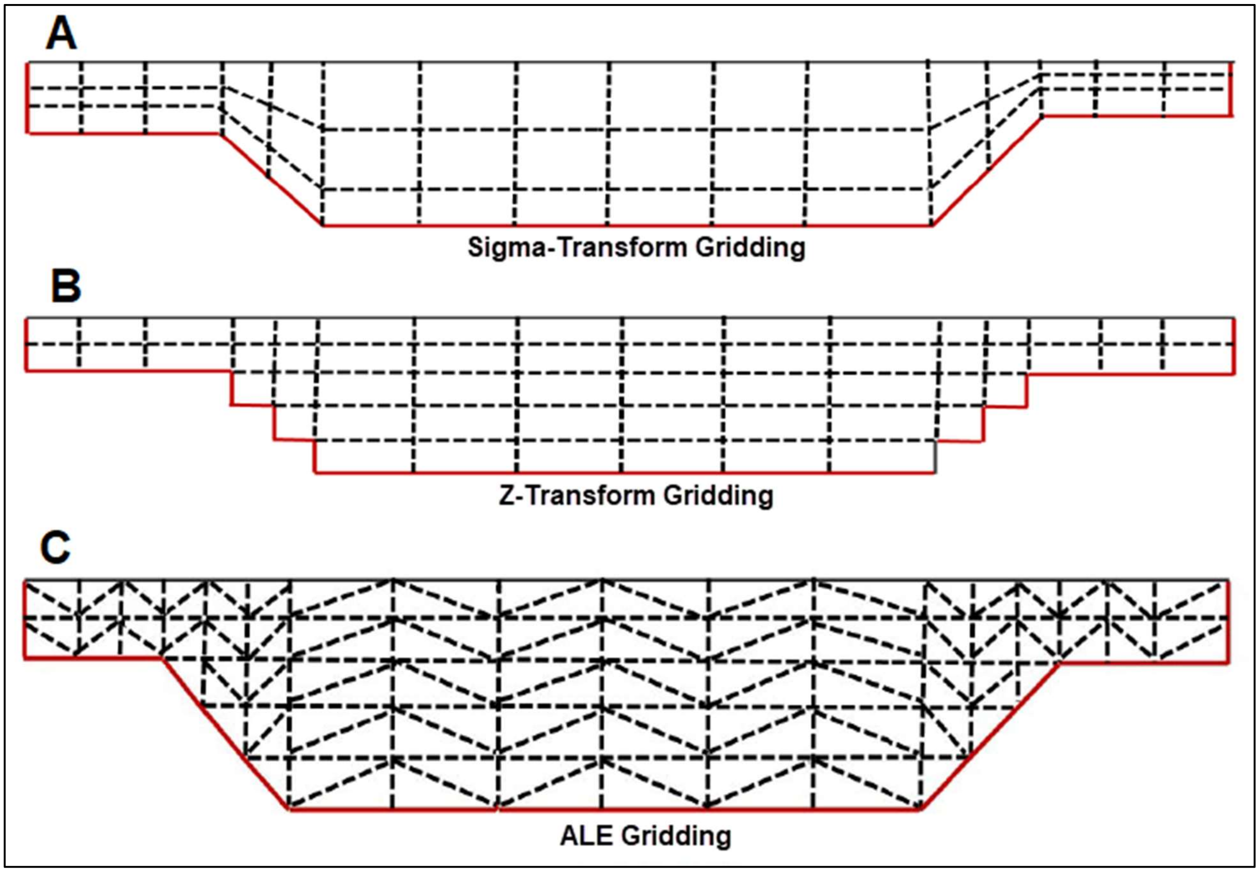

Figure 7-17. Meshing Strategies for 3D.

A Brief Note on 3D-Non Hydrostatic (3D-NH) Meshing

Meshing for 3D-NH models is significantly different from meshing for 3D-Hydrostatic models. 3D-NH models are in general not restricted to vertical columns and can have a fully unstructured mesh layout. This is due to the fact that 3D-NH models are solving for the pressure instead of assuming a hydrostatic distribution.

Model Discretization

In general, 2D and 3D models do not require knowledge of the exact flow paths of the water before laying out the model. However, knowledge of the flow paths can be very useful in developing a more detailed mesh for 2D and 3D modeling. For example to accurately model flow of salinity in and out of an estuary, information about location of stratification must be known *apriori* in order to provide enough vertical resolution to capture this stratification, and to avoid vertical resolution in areas that do not undergo this stratification. 3D models are computationally expensive, and efficiency in resolution is essential.

Once the model extents are defined, the basics of a 3D model development are similar to the 2D model in the horizontal resolution. Increased resolution in terms of smaller cell/element size is required for locations of interest, or locations where there is a rapid change in hydrodynamics or transport. These regions often include structures, bends, navigation channels, inflow locations, contractions or expansions and the like.

Irrespective of rapidity of change, 3D models require that the horizontal resolution be enough to capture bathymetric features essential to simulate the 3D flow/transport field. Figure 7-18 provides an example of this increased resolution in a location where a navigation channel splits into two, and increased resolution is required to provide adequate vertical resolution when layering information is generated.

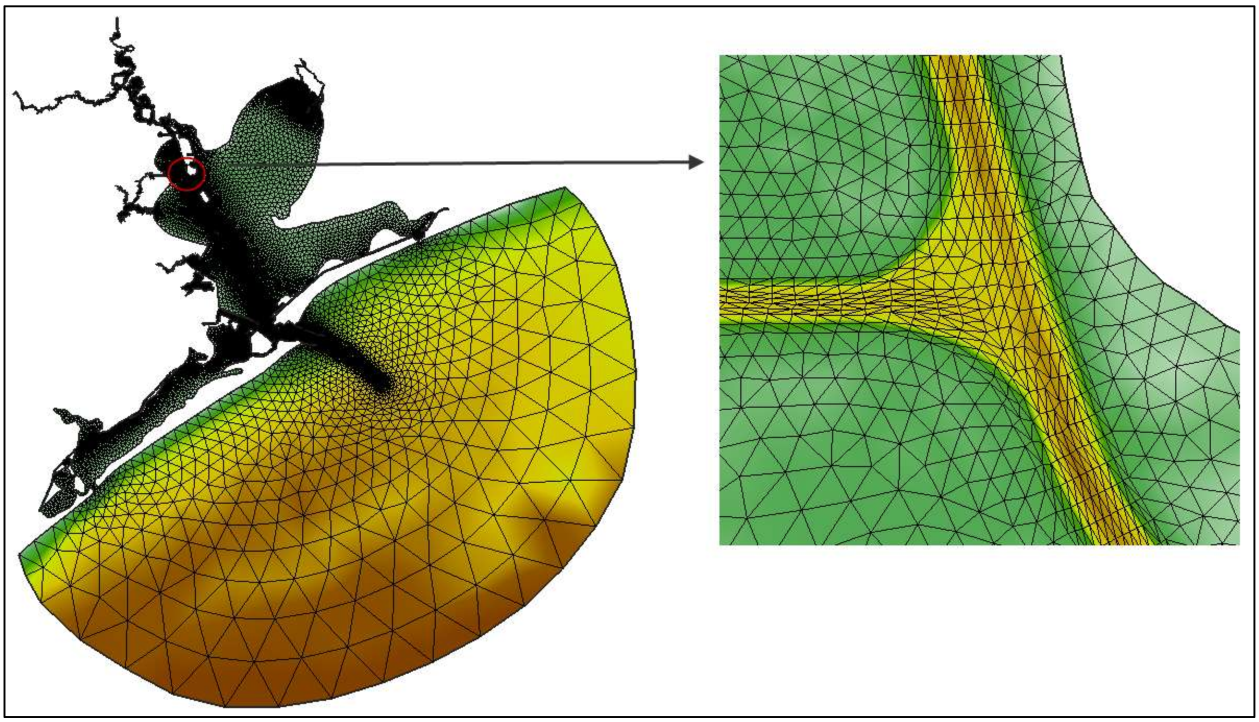

Figure 7-18. Horizontal meshing for 3D models, green to red color indicates the bathymetric features (deeper to shallower).

In addition to flow features of interest, computational resources available have an influence on the horizontal resolution as well. This influence is especially true for 3D hydrostatic codes; consequently, these 3D models require that a columnar structure be maintained, and every horizontal cell/element will require a vertical layering description. However, an un-necessarily high horizontal resolution will increase computation costs as well as run-times for the 3D simulation.

The 3D vertical layering information, and the manner in which the layers are laid out, greatly, depends upon the type of coordinate description used, and the nature of the simulation. For example, if the σ transformation is used, then the number of layers in the vertical will be carried out throughout the simulation extents and smoothing of any abrupt slope changes or discontinuities in bathymetry will be required. However, if z transformation is used then the vertical spacing of the layers has to be determined (keeping in mind that this transformation modifies the bathymetry to stair-step). Otherwise, if an ALE 3D meshing is used, then the vertical layering must maintain nodes at constant contour levels. An example of ALE layering is presented in Figure 7-19, note that in a layer the nodes are laid out at approximately the same contour level.

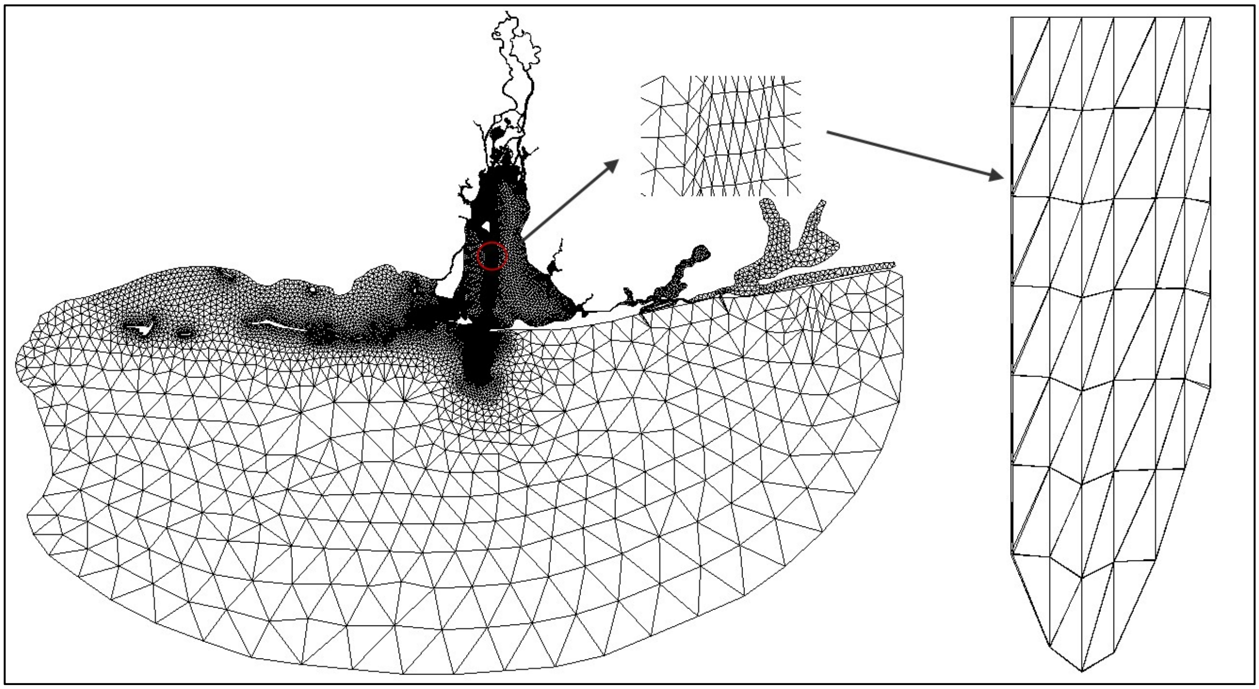

Figure 7-19. Example of Arbitrary-Lagrangian Eulerian (ALE) vertical meshing for 3D models.

Defining Roughness Coefficients

Defining friction values (roughness coefficients, roughnes height etc.) is very similar between 2D and 3D. Roughness coefficients are, in general, defined based on available data such as land-use/land-cover, vegetation type, sediment behavior etc. The initial 3D mesh/grid must be built taking into account the various types of roughness domains/materials/regions that might be in the modeled domain, this is essential as creating additional roughness domains/materials/regions in a 3D grid/mesh is a time consuming, and tedious process. Once the initial 3D simulations are completed, the calibration and validation process can be used to guide the modification of the initial roughness values where required.

Defining Turbulence Closure, Background Eddy Viscosity, and Diffusion Values

3D models, when compared to 2D models, provide robust means to compute turbulence closure values. These turbulence closure values are used to account for processes that arise as a result of chaotic changes and/or gradients in pressure and velocity. Turbulence increases hydraulic or hydrodynamic losses in the flow field, most often through the generation of vortices that may or may not interact with each other. In 2D and 3D numerical modeling, turbulence closure models are used to account for losses in the flow field at scales smaller than the mesh/cell resolution. The mathematical description of turbulence closure schemes is involved, and the interested reader is guided to Savant (2015), Blumberg et al., (1992), Mellor and Yamada (1982), and Henderson-Sellers (1982) for an in-depth discussion. Table 7-1, presents general guidelines for the utilization of turbulence closure in 3D models.

Table 7-1. Suggested Turbulence Closure Schemes

Flow Condition	Vertical Turbulence Closure	Horizontal Turbulence Closure
Stratified Estuarine Flow for Micro Tidal Systems	Mellor-Yamada Level 2	Smagorinski
Stratified Estuarine Flow for Macro Tidal Systems	Mellor-Yamada Level 2.5, k-ε	Mellor-Yamada Level 2.5, k-ε, Smagorinski
Riverine Flows with no Hydraulic Structures	Mellor-Yamada Level 2	Smagorinski
Riverine Flows with Hydraulic Structures	Mellor-Yamada Level 2.5, k-ε	Mellor-Yamada Level 2.5, k-ε, Smagorinski
Reservoirs with Selective Withdrawal	k-ε	k-ε
Stratified Reservoirs/Lakes	Mellor-Yamada Level 2	Smagorinski

Higher level or 2-equation turbulence closure models such as Mellor-Yamada Level 2.5, and k-ε involve the solution of transport equations for the generation and dissipation of turbulence. As a consequence, the computation cost is increased when compared to 1-equation models such as Mellor-Yamada level 2, and other simpler models.

Most 3D numerical models provide the option to provide background values for the eddy viscosity tensor. These background values are often simulation and system dependent, and the values are difficult to ascertain. As a consequence these values provide a convenient method to stabilize the numerical simulation, and as calibration tools. It is recommended that these values be kept at levels that stabilize the simulation, but no higher unless justified by observations.

For 3D simulations that involve the transport of constituents, the specification of turbulent and background diffusion values is required. Moreover, the suggestions provided in Table 7-1 are valid for turbulent diffusion as well. Transport equations in 2D as well as 3D incorporate diffusion, turbulent as well as background, as terms that redistribute mass. As a consequence, extreme care must be exercised when specifying background diffusion values. Background diffusion values are extremely tedious to ascertain and are user specified, and therefore provide a convenient way to match observations. Utilizing background diffusion values in this manner, might allow the simulation to match the values observed under one condition, but will in all likelihood fail the validation under a different set of flow conditions. Therefore, for a prognostic model, background values must not be used as calibration tools, and should be set at a value that stabilizes transport but no higher unless justified by observations.

Hydraulic and Other Structures

Defining and modeling hydraulic structures is similar in 2D and 3D modeling approaches, with the only difference being in the vertical layering structure and the presence of turbulence computations that inherently account for expansion and contraction losses. Unpressurized structures such as unsubmerged bridges, culverts etc. can be directly modeled using the equations of motions; for pressurized structures additional pressure considerations and flow splits must be accounted for.

When modeling hydraulic structures, 3D models can provide more hydraulic information for bridge pier scour and other phenomenon when compared to 2D simulations. However, due to the presence of vertical accelerations 2D and hydrostatic 3D models are inappropriate for near-field information of flow, or transport such as sediment. Instead, for these simulations, 3D-Non Hydrostatic models should be utilized.

Boundary Conditions

The specification of boundaries are similar in 2D and 3D models. 2D models in general specify boundaries on external edges/cells and internal edges/cells as either inflows, outflows, stages, precipitations, evaporation and infiltration. 3D models allow for the specification of the same boundaries, but these specifications can be depth varying if desired. Supercritical flow boundaries, common in 2D models, are not utilized in 3D models due to extreme deformations of the surface. Baroclinic transport, such as salinity and temperature, can cause 3D boundaries to have flows in opposite directions. This type of transport is commonly observed at locations with water surface elevation boundaries.

Model Calibration

The model calibration process is one of the most important steps in the development of a hydraulic model. Calibration of any hydraulic model is required in order to understand if the model is capable of reproducing past flows/transport, and if it has any chance of predicting future flow/transport events. The calibration process also allows for greater understanding of the models sensitivity to the data, friction forces, and other empirical coefficients. A model that is not calibrated is truly just a numerical experiment. Indeed, an uncalibrated model may or may not even be close to reproducing realistic water surface elevations, flows and transport throughout the system. Uncalibrated 3D models should never be used to make engineering decisions, even in emergency situations. This assertion is due to the fact that the effort needed to create a 3D model will generally require time scales longer than what the emergency will allow, and the complexity of 3D modeling lends itself to making errors.

The process of calibrating a model is very similar between the 2D and 3D modeling approaches; however, 3D model calibration can be more difficult and time consuming. The data required to perform a model calibration is mostly the same, except more detailed calibration of 3D models requires detailed velocity distribution measurements to ensure the accuracy of the vertical velocity profiles. For density dependent 3D flows, calibration of transport is also required and can be very complex. This calibration complexity is because of feedback interactions between hydrodynamics and transport (i.e., the hydrodynamics influence the transport, and the transport in return influences the hydrodynamics through density impacts).

Data required to calibrate and validate a 3D model is:

- Water surface elevations,
- Horizontal and vertical velocity profiles,
- Flow splits, and
- Vertical profiles of transported constituents.

Time and Cost Issues

The time to develop a model with the 2D or 3D modeling approach can vary, depending on the type and purpose of the model. For example, if a "Quick and Dirty" model needs to be developed during a flood emergency, then it is much faster to lay out a 2D flow area polygon, set a basic cell/element size, attach some boundary conditions, and compute. On the other hand, 3D models can be extremely time consuming and tedious to create. The creation of a 3D model requires the creation of a 2D domain as well as vertical layering information, and are therefore always more time consuming to create. Estimating the base parameters, such as: roughness coefficients, background parameters and other empirical coefficients needed for each of the modeling approaches requires about the same amount of time.

In general, the computational cost for 2D models, which solve the continuity, x-, and y-momentum equations for u, v, and h (i.e., 3 degrees of freedom per computational cell/element) is less than 3D models. This computational cost difference is because 3D models solve the continuity, x-, y-momentum and the hydrostatic assumption to solve for u, v, w, and h (i.e., 4 degrees of freedom per computational cell/element); and therefore are always more computationally expensive to solve when compared to 2D. The addition of the vertical layering further increases the computational expense.

Recent advances in 2D models have allowed the simulation of decadal scale and continental scale hydrodynamics; however, even the most advanced 3D models are limited to yearly scales and significantly smaller spatial scales of the order of tens of miles.

On the other hand, 3D models require hardware computationally more powerful than those for 2D models. Therefore, more often than not, 3D models are run on multi-processor systems such as high performance computers (HPC), or cloud computing. Desktop machines can be utilized for relatively small 3D models depending upon the model resolution, cell/element count, and the spatial/temporal scales of simulation.

Depending upon the type of model, 3D models can provide output for depth, pressure, velocities, concentrations etc. Consequently, this output can be of the order of gigabytes, if not larger, and is complex to process. However, the visualization tools for 3D models are primitive when compared to the visualization tools for 2D models. As a consequence, engineers create ad-hoc ways to extract data generated by 3D models, and/or extract layer information and visualize each layer or column as 2D. Such ad-hoc practices adds an additional time cost in post-processing of results.

Alternatively, the computational and time cost of running a 3D model can be reduced if a 2D model is used to "*hot start*" the 3D model, but this requires the existence of 2D results or the creation of a 2D simulation.

Modeler Knowledge, Skills, and Abilities

Whether performing 2D or 3D modeling, the modeler should have a good background in hydraulics/hydrodynamics, hydraulics around structures, transport, and numerical solutions of non-linear equations. Additionally, no matter what computer model and modeling approach is selected, the modeler should take classes on how to use that specific piece of software effectively for 2D and/or 3D modeling. Self-study is great to get started on using 2D models, but the additional knowledge gained from taking a class from the software developers/experts in using the software will often prove to be invaluable. In contrast, 3D models are extremely complex, and self-study is of lesser use. This discrepancy is because even though all 3D models solve the same equations of motion and transport, the discretization in the horizontal and vertical, and good practices are model specific. In addition the application and utilization of turbulence options vary between models, and within the same model depending upon certain conditions. Therefore, the modeler must familiarize oneself with techniques to pre- and post-process the 3D mesh/grid to satisfy good layering practices for the model being used, and to visualize simulation results.

Moreover, the modeler needs to perform detailed studies of systems in which they have good bathymetric, structure information, and historic data to perform model calibrations. If the modeler is new to modeling, they should always seek out guidance and assistance from experienced modelers. Additionally, all models should be reviewed by independent experts in order to ensure that the work performed was take with an appropriate modeling choice (2D or 3D); applied correctly; contains the appropriate amount of detail for the given location and study type; was calibrated effectively, and is reproducing reasonable hydraulic results for the events and alternatives being modeled.

Summary of 2D and 3D Modeling Advantages and Disadvantages

There are definite advantages and disadvantages to both the 2D and the 3D modeling approaches. This section of the document summarizes the advantages and disadvantages that have been described previously in this document. The discussion is based on advantages and disadvantages from the perspective of 2D modeling versus 3D modeling. Even though 2D modeling advantages and disadvantages were listed previously, they'll be presented here from the perspective of comparison to 3D modeling.

2D Modeling Advantages:

- The flow path of the water, for all events, does not have to be known to develop the model. However, the extent of the domain does need to be correctly defined.
- The direction of the flow can change during the event. Water can move in any direction, based on energy and momentum of the flow.

- Velocity, momentum, and the direction of the flow are more accurately accounted for. In fact, this accuracy is especially true for flow going over roads, levees, barriers, structures, around bends, and at flow junctions/splits. Additionally, 2D models can be used to analyze 2D eddy zones within the flow field. Around bends, 2D models produce accurate water surface elevations, but velocity distributions might be erroneous due to the existence of helical flow.
- Energy and force losses due to contractions and expansions, etc. are directly accounted for, and do not require empirical coefficients, increased roughness, or user defined ineffective flow areas.
- The mapping of the inundated area, as well as velocities, and flood hazards (depth x velocity) is accurate.
- Detailed modeling of hydraulic structures, in a full 2D modeling approach, can provide more insight into the flow distribution approaching, going through, and coming out of a structure.
- Fast computation times, as well as time to results.

2D Model Disadvantages:

- A false sense of confidence in 2D model results is possible for scenarios where 3D effects are not obvious, such as flow around bends, flow around structures, reservoir/lake simulation, etc.
- No information about the vertical structure of the flow can be obtained from 2D models.

3D Modeling Advantages:

- 3D models provide a better representation of the hydraulics/hydrodynamics in a system.
- 3D models provide the only way to realistically model hydraulic systems that stratify, and the resultant transport. These include systems such as reservoirs, lakes and estuaries.
- 3D models can provide accurate velocity behavior for flow around bends, and accurately replicate resultant sediment, or other transport behavior.
- 3D models provide a better representation of hydraulics around structures. However, to accurately represent hydraulics around structures 3D-Non Hydrostatic models must be used.
- 3D models provide accurate hydraulic behavior for hydraulic systems that include selective withdrawal.
- Energy losses in the form of eddies, expansion/contraction losses, as well as losses due to submerged obstructions such as weirs, can be accurately replicated by 3D models.
- In some ways, 3D models are more physically based than 2D models. Parameters such as eddy viscosity coefficients that are user specified in 2D models, are computed internally by 3D models.

3D Modeling Disadvantages:

- The flow path of the water, for all events, have to be known to develop the vertical layering structure in the model for 3D hydrostatic models.
- The process to acquire adequate data such as, background viscosity etc., for 3D modeling requires field testing/data collection.
- The development of a 3D mesh/grid usually requires the initial description of the domain in 2D.
- Mesh development in a 3D model can be complicated, and may require iterations.
- The complex nature 3D models increase the probability of instabilities.. This instability requires in-depth knowledge of the code and may require frequent restarts to the model, either through hot starting or by modifying the parameters.
- The computation time and cost required for 3D can be extreme and prohibitive.
- Post-processing of results to obtain usable data is required. Visualization of mesh/grid and results in 3D may require additional training and knowledge of post-processing software.

The decision between the choices of 2D or 3D models must maintain fidelity to the goals of the project being investigated, and the principle of “simplest, and technically sound path to achieve the goals”. For example, if flood extents are of interest for a river that includes river bends or other 3D features, a 2D model might be sufficient even though velocity distribution is incorrect; on the other hand, if sediment behavior is of interest a 3D model might be more appropriate. Another possibility is that the same system might require a 2D model or a 3D model depending upon the hydraulics being investigated. The engineer must avail oneself of the appropriate knowledge of the system to successfully attain goals of the project.

Chapter 8

Physical Hydraulic Models

Steady improvements in numerical techniques and computer technology over the past half-century have greatly expanded the range of practical numerical modeling applications. Nonetheless, there are many practical hydraulic engineering problems for which scaled physical hydraulic models continue to be the only or the most cost-effective solution method. The digital revolution that enabled advances in numerical modeling has also expanded options for precisely controlling and measuring flows and related phenomena in scaled models. Thus, the range of potential applications and fidelity of physical hydraulic models are also expanding. Additionally, there are problems for which a combination of numerical and physical modeling (i.e., hybrid modeling), may be appropriate.

This document does not present detailed guidance on the development, usage, and limitations of scaled physical hydraulic models. Consultation with an appropriate subject matter expert is recommended early in the study process for any project where the application of a scaled physical model may be appropriate.

General areas of applicability include:

- Flows internal to hydraulic structures, including:
- Lock filling and emptying systems
- Outlet works
- Hydropower units
- Determination of forces acting on structures
- High velocity channels (particularly for complex channel/structure geometry or flows near critical depth)
- Shallow draft navigation in the vicinity of structures
- Local scour protection (particularly spillway design)
- Scour around atypical bridge piers
- Fish passage
- Pump intakes (as specified by formal standards)

Scaled physical models may be classified as process models, used to investigate generic physical processes, or as design models, used to investigate specific aspects of a proposed alternative or an experienced failure. Process models are a critical source of information for scientific advances in the fields of environmental fluid mechanics, sedimentation, river engineering, etc. and for development and verification of numerical models. Design models are a widely accepted tool for exploring design alternatives and optimizing cost and performance while reducing uncertainty. Design models can also provide validation data for scaled numerical models and thus reduce uncertainty in prototype scale numerical models. For hydraulic structures and other complex hydraulic systems, the construction, operations and maintenance cost savings and benefits from performance improvements can exceed the cost of a design model by significant margins (ASCE, 2000).

Chapter 9

Summary

Numerical modeling of natural as well as artificial hydraulic systems has been an integral part of management since the advent of modern computing. Early days of numerical modeling saw the creation and wide utilization of simplified numerical models that were restricted to computing coarse scale properties of flow such as residence times based on flow and volume. As computational resources became available the numerical models increased in complexity to include multi-dimensional simulations. These simulations were relegated to small time and spatial scales, of the order of hours and miles respectively. It wasn't until the availability of robust 1D models that numerical modeling of hydraulic systems became routine, and widespread enough to be used in planning and management.

Computational resources underwent a revolution in the last decade of the 20th century, and multi-dimensional numerical modeling became a realistic enabler for designers, planners and managers alongside 1D modeling. Recent advances in high performance computing (HPC), graphical processing units (GPU), cloud computing, as well as desktop computing has allowed multi-dimensional numerical simulations of hydraulic systems in detail. The availability of 1D, and multi-dimensional (2D, and 3D) has led to the question of "which ones to use?" or more appropriately "which ones to not use?"

The choice of the kind of numerical model, more often than not, is dependent upon the skill, experience, knowledge and preference of the modeler. A skilled modeler can glean useful insights from all kinds of numerical models, however the judicious selection of the appropriate numerical model holds the promise of significant time and cost savings. In addition, the choice of a numerical model is dependent on the purpose for which the system is under investigation. The same system might require a different numerical model based on project goals, the time frame, computational resources, as well as data requirements. For example the determination of inundation extents can be performed with 1D, 2D, as well as 3D models, but time and computational costs may indicate that a 1D model should be used because 2D/3D models will be relatively time and computation prohibitive. The same system, if investigated for detailed velocities, will require the utilization of a 2D/3D model instead of the 1D model.

The unavailability of data is usually a poor excuse to deter the use of a model if the goals, and physics of the system indicate that a 2D or a 3D model is required. In such cases, additional data must be acquired to accurately simulate the system, and to achieve the goals of the project. This document has presented guidelines, and suggestions about where the various numerical techniques are valid, and to what extent. Table 9-1 presents various systems that are commonly simulated by hydraulic engineers, and the simplest type of model to use.

Table 9-1. Recommended modeling for various commonly modeled systems.

System	Purpose	1D	2D	3D	3D-NH / 3D-NHMP	Use
River	Flood Extents	✓	✓ (SW2, DW)			P, E, D
River	Velocities	✓	✓ (SW2)			P, E, D
River	General Transport* Behavior	✓	✓ (SW2)			P, E, D
River	Detailed Transport* Behavior		✓ (SW2)	✓		P, E, D
River (with bends)	Flood Extents	✓	✓ (SW2, DW)			P, E, D
River (with bends)	Velocities/Transport*		✓(SW2 with corrections)	✓		2D (P, E) 3D (P, E, D)
Supercritical Flows	Flood Extents	✓	✓(SW2)			P, E, D
Supercritical Flows (with bends)**	Flood Extents/Velocities/Transport*		✓(SW2 with corrections)	✓	✓	2D (P, E) 3D (P, E) 3D-NH/NHMP (P,E,D)
Dam Break/Breach	Flood Extents	✓	✓(SW2)			P, E, D
Levee Break/Breach	Flood Extents		✓ (SW2, DW)			P, E, D
River (with structures)	Flood Extents	✓	✓ (SW2)			P, E, D
River (with structures)	Velocities/Transport*		✓(SW2)	✓		2D (P, E) 3D (P, E)
Reservoirs/Lakes	Residence Time		✓	✓		P, E, D
Reservoirs/Lakes	Transport*			✓		P, E, D
Estuaries (Well Mixed)	Water Surface/Velocities/Transport*		✓(SW2)			P, E, D
Estuaries (Stratified)	Water Surface/Velocities/Transport*			✓		P, E, D
Gates and Spillways**	Water Surface/Velocities		✓ (SW2)	✓	✓	2D/3D (P,E) 3D-NHMP (P,E,D)

Note: DW = Diffusive Wave, SW2 = 2D Shallow Water, 1D = One-Dimensional, 2D = Two-Dimensional, 3D = 3D Hydrostatic, 3D-NH = 3D-Non Hydrostatic, 3D-NHMP = 3D-Non Hydrostatic Multi Phase, P = Planning, E = Evaluation, D = Design, Transport = Including sediment, ** = Physical Model Recommended*

References

American Society of Civil Engineers. 2000. *Hydraulic modeling: concepts and practice*, ASCE Manuals and Reports on Engineering Practice No. 97. Environmental and Water Resources Institute, ASCE, Reston, VA.
https://doi.org/10.1061/9780784404157

Bernard, R. S., and Schneider, M. L. 1992. *Depth-averaged numerical modeling for curved channels.* Technical Report HL-92-9, U.S. Army Engineer Waterways Experiment Station, Hydraulics Laboratory, Vicksburg, MS.
http://hdl.handle.net/11681/13151

Blumberg, A. F., B. Galperin, and D. J. O'Connor. 1992. *Modeling vertical structure of open-channel flows.* Journal of Hydraulic Engineering Vol. 118, Issue 8, 1119–1134.
https://doi.org/10.1061/(ASCE)0733-9429(1992)118:8(1119)

Brown, G.L. 2012. *A Quasi-3D Suspended Sediment Model Using a Set of Correction Factors Applied to a Depth Averaged Advection Diffusion Equation.* IIHR Third International Symposium on Shallow Flows, Iowa City, IA, June 4-6, 2012.

Chow, V.T. 1959. *Open Channel Hydraulics*, McGraw-Hill, New York, NY.

Ellet, C. (1853). *The Mississippi and Ohio Rivers: containing plans for the protection of the delta from inundation and improving the navigation of the Ohio and other rivers by means of reservoirs*. Report to Congress, Lippincot, Grambo, and Co., Philadelphia, PA.

Finne, J., B. Donnell, J. Letter, and R. S. Bernard. 1999. *Secondary flow correction for depth-averaged flow calculations*. Journal of Engineering Mechanics. Vol. 125, Issue 7, 848-863.
https://doi.org/10.1061/(ASCE)0733-9399(1999)125:7(848)

Hydrologic Engineering Center (HEC). 2016. HEC-RAS *River Analysis System, 2D Modeling* User's Manual, Version 5.0, CPD-68A. U.S. Army Corps of Engineers, HEC, Davis, CA.

Hydrologic Engineering Center (HEC). 2016. HEC-RAS *River Analysis System*, Applications Guide, Version 5.0.4, CPD-70. U.S. Army Corps of Engineers, HEC, Davis, CA.

Hydrologic Engineering Center (HEC). 2016. HEC-RAS *River Analysis System*, Hydraulic Reference Manual, Version 5.0, CPD-69. U.S. Army Corps of Engineers, HEC, Davis, CA.

Hydrologic Engineering Center (HEC). 2016. HEC-RAS *River Analysis System*, User's Manual, Version 5.0, CPD-68. U.S. Army Corps of Engineers, HEC, Davis, CA.

Henderson-Sellers, B. 1982. *A simple formula for vertical eddy diffusion coefficients under conditions of non-neutral stability*. Journal of Geophysical Research: Oceans, Vol. 87, Issue C8, 5860–5864.
https://doi.org/10.1029/JC087iC08p05860

Mellor, G. L., and T. Yamada. 1982. *Development of a turbulence closure model for geophysical fluid problems*. Reviews of Geophysics and Space Science, Vol. 20, No. 4, 851–875.
https://doi.org/10.1029/RG020i004p00851

Savant, G. 2015. *Three-Dimensional Shallow Water Adaptive Hydraulics (ADH-SW3): Turbulence Closure*. CR-15-1. U.S. Army Corps of Engineers, Engineer Research and Development – Coastal and Hydraulics Laboratory, ERDC/CHL, Vicksburg, MS.
http://hdl.handle.net/11681/1801

Jeremy A. Sharp, Tate. O. McAlpin, Ronald E. Heath, Gary C. Lynch, and Howard E. Park. 2013. *2D Hydrodynamic Investigation of Olmsted Cofferdams*. TR-13-6. U.S. Army Corps of Engineers, Engineer Research and Development – Coastal and Hydraulics Laboratory, ERDC/CHL, Vicksburg, MS.
http://hdl.handle.net/11681/7342