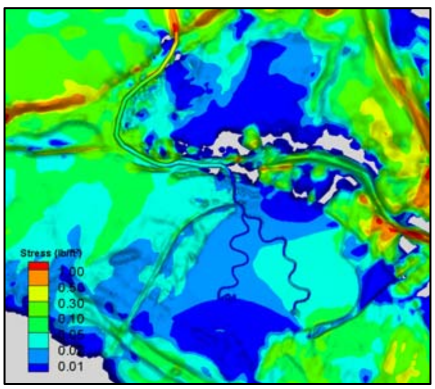
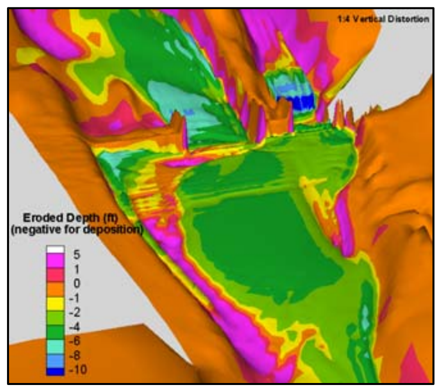
RECLAMATION

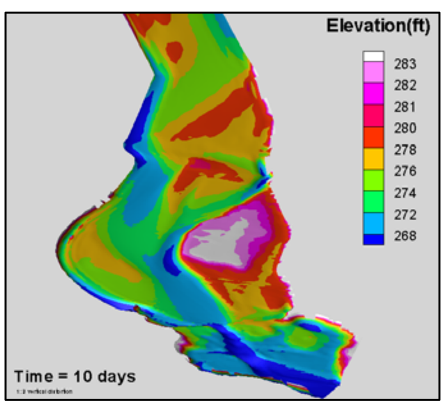
Managing Water in the West

SRH-2D version 2: Theory and User's Manual

Sedimentation and River Hydraulics – Two-Dimensional River Flow Modeling







U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Denver, Colorado

November 2008

(Intentionally blank)

SRH-2D version 2: Theory and User's Manual

Sedimentation and River Hydraulics – Two-Dimensional River Flow Modeling

Prepared by

Yong G. Lai, Ph.D., Hydraulic Engineer

Bureau of Reclamation
Technical Service Center
Sedimentation and River Hydraulics Group

U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Denver, Colorado

November 2008

Mission Statements

The mission of the Department of the Interior is to protect and provide access to our Nation's natural and cultural heritage and honor our trust responsibilities to Indian Tribes and our commitments to island communities.

The mission of the Bureau of Reclamation is to manage, develop, and protect water and related resources in an environmentally and economically sound manner.

Acknowledgments

A number of researchers and engineers have made contributions to the review and testing of SRH-2D version 2, along with the documentation. Their effort has greatly enhanced the quality of the work reported. In particular, the following individuals are acknowledged: Timothy Randle, Blair Greimann, Robert Hilldale, Jennifer Bountry, and Victor Huang at the Technical Service Center of the Bureau of Reclamation (Denver, CO), David Mooney at the Mid-Pacific Region of the Bureau of Reclamation (Sacramento, CA), and Chih Ted Yang at the Colorado State University (Fort Collins, CO). Peer review of this document was performed by Robert Hilldale.

The work reported was funded by a number of sources, including the Environmental Protection Agency (EPA) under the Interagency Agreement No.DW14948044, Reclamation's Science and Technology Program, and a number of Reclamation projects.

Disclaimer

CONTENTS

SUMMARY 1

CHAPTER 1 3

INTRODUCTION 3
1.1 General 3
1.2 Modeling Concept and Capabilities 4
1.3 Limitations 7
1.4 Acquiring SRH-2D 7
1.5 Disclaimer 7

CHAPTER 2 9

GETTNG STARTED 9
2.1 Model Structure 9
2.1.1 Mesh Generation Program 9
2.1.2 SRH-2D Package 10
2.1.3 Post-Processing Program 11
2.2 Modeling Steps 11
2.2.1 Mesh Generation 12
2.2.2 Preprocessing 12
2.2.3 Main Solver Execution 13
2.3 SRH-2D Output Files 14
2.3.1 _RES File 14
2.3.2 _OUT File 14
2.3.3 _RST*n* File 14
2.3.4 Result Output File 14

CHAPTER 3 15

FULL-INTERFACE MODE WITH SMS 15
3.1 SRH-2D Template File 15
3.1.1 Global Parameters Setup 17
3.1.2 Boundary Condition Setup 20
3.1.3 Manning's Roughness Coefficient 23

CHAPTER 4 25

PARTIAL-INTERFACE MODE: INPUT COMMANDS 25

CHAPTER 5 35

TUTORIAL 35
5.1 A Subcritical Flow in a Channel 35

CHAPTER 6 39

GOVERNING EQUATIONS 39
6.1 Flow Equations 39

CHAPTER 7 43

INITIAL AND BOUNDARY CONDITIONS 43
7.1 Initial Conditions 43
7.2 Inlet Boundary 43
7.3 Exit Boundary 44
7.4 Solid Wall Boundary 45
7.5 Symmetry Boundary 46

CHAPTER 8 47

NUMERICAL METHODS 47
8.1 Flow Solver 47
8.1.1 Discretization 47
8.1.2 Side Normal Velocity Calculation and Elevation Correction Equation 50
8.1.3 Summary of Solution Procedure 52

CHAPTER 9 53

VERIFICATION CASES 53
9.1 1D Subcritical Flow in a Channel 53
9.2 1D Transcritical Flow in a Channel 55
9.3 2D Diversion Flow in a Channel 57

CHAPTER 10 61

APPLICATION CASES 61
10.1 Savage Rapids Dam Removal Study 61
10.1.1 Topography and Mesh 61
10.1.2 Case Modeled, Boundary Conditions, and Other Parameters 63
10.1.3 Comparison of Water Surface Elevation 63
10.1.4 Comparison of Velocities and Flow Patterns 64
10.2 Study of Sandy River and Columbia River Interaction 67
10.2.1 Solution Domain, Mesh, and Flow Roughness 68
10.2.2 Input Data 72
10.2.3 Comparison of Water Surface Elevation 73
10.2.4 Comparison of Flow Velocity 75
10.3 Other Application Cases 82

REFERENCES 85

APPENDIX A 89

ON MESH GENERATION USING SMS 89

APPENDIX B 93

INPUT AND OUTPUT FORMATS 93
B.1 SRH Formats 93
B.2 TECPLOT Format 93
B.3 XMDF Format 93
B.4 GENERIC Format (not supported currently) 93

APPENDIX C 95

DYNAMIC INPUT FILE (DIP) 95

APPENDIX D 97

FORMAT OF TIME SERIES FUNCTION AND GENERAL FUNCTION 97

APPENDIX E 99

COMMON ERRORS 99

APPENDIX F 101

SPECIAL TREATMENT IN MESH ZONES 101

TABLES

Table 1. Calibrated Manning’s Coefficients in Different Zones Shown in Figure 33.. 72

FIGURES

Figure 1. Illustration of zonal partition and mesh layout....................................... 4

Figure 2. A sample SRH-2D mesh: quadrilateral cells are used along the main channel and levees but mixed coarser cells are in the floodplains 5

Figure 3. A sample SRH-2D mesh that uses a combination of structured quadrilateral cells and unstructured mixed-shape cells 6

Figure 4. The pop-up window to append the template file to SMS executable.... 16

Figure 5. An 80-by-3 mesh used for simulation for test 1 of MacDonald (1996) 36

Figure 6. A sample window session running ***srh2d20*** for the tutorial case 1....... 37

Figure 7. Schematic illustrating a polygon P along with one of its neighboring polygons N.. 48

Figure 8. An 81-by-4 mesh used for simulation for test 1 of MacDonald (1996) 53

Figure 9. Comparison of simulated water surface elevation with analytical solution for test case 1 of MacDonald (1996).. 54

Figure 10. Comparison of simulated water depth with analytical solution for test case 1 of MacDonald (1996)... 54

Figure 11. 3D view of bed elevation and simulated water surface elevation for test case 6 of MacDonald (1996)... 56

Figure 12. Comparison of simulated water surface elevation with analytical solution for test case 6 of MacDonald (1996).. 56

Figure 13. Comparison of simulated water depth with analytical solution for test case 6 of MacDonald (1996)... 57

Figure 14. Part of the quadrilateral mesh used for simulation of the diversion flow 58

Figure 15. Comparison of water surface elevation along both walls of the main channel for the Shettar and Murthy (1996) case.. 59

Figure 16. Comparison of water surface elevation along both walls of the side channel for the Shettar and Murthy (1996) case.. 59

Figure 17. Comparison of x-velocity (U) profiles at selected x locations in the main channel for the Shettar and Murthy (1996) case................................. 60

Figure 18. Comparison of y-velocity profiles at selected y locations in the side channel for the Shettar and Murthy (1996) case. 60

Figure 19. Plainview and bed elevation contours of the simulated area for the Savage Rapids Dam removal project 62

Figure 20. A Perspective View of the Topography of the Modeled River Reach. 62

Figure 21. Comparison of Predicted and Measured Water Surface Elevations.... 64

Figure 22. Velocity Measurement Points for the Simulated River Reach (Points are Shown in Red) 65

Figure 23. Comparison of Predicted and Measured Velocity Vectors at Cross Sections 1 to 4 66

Figure 24. Comparison of Predicted and Measured Velocity Vectors at Cross Sections 5 to 8 66

Figure 25. Comparison of Velocity Vectors and Flow Patterns downstream of the Dam 67

Figure 26. Aerial photo of the study area at the Sandy River Delta 68

Figure 27. Solution domain for the Sandy River Delta simulation. West (left) side of the Columbia River is the exit boundary, east (right) side is the inlet boundary, and south (bottom) side is the inlet boundary of the Sandy River 69

Figure 28. Mesh for the Sandy River Delta project: entire solution domain. 69

Figure 29. Mesh for the Sandy River Delta project: the Sandy River Delta area. 70

Figure 30. Mesh for the Sandy River Delta project: Dam area. 70

Figure 31. Contour plot of the bed elevation for the Sandy River Delta project.. 71

Figure 32. 3D perspective view of the topography for the solution domain. 71

Figure 33. Roughness zones used for the Sandy River Delta Project.................. 72

Figure 34. Comparison of simulated and field-measured water surface elevations along the Sandy River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D) 74

Figure 35. Comparison of simulated and field-measured water surface elevations along the Columbia River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D) 75

Figure 36. Comparison of simulated and field-measured velocity magnitudes along the Sandy River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D) 76

Figure 37. Comparison of simulated and field-measured velocity magnitudes along the Columbia River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D) 77

Figure 38. Seven regions (blue boxes) used for velocity vector comparison; Red points are the locations where velocity measurements were made............ 77

Figure 39. Comparison of velocity vectors in Region 1 (GSTAR-W is the former name of SRH-2D) 78

Figure 40. Comparison of velocity vectors in Region 2 (GSTAR-W is the former name of SRH-2D) 78

Figure 41. Comparison of velocity vectors in Region 3 (GSTAR-W is the former name of SRH-2D) 79

Figure 42. Comparison of velocity vectors in Region 4: Left is upstream and right is downstream portion of the region (GSTAR-W is the former name of SRH-2D) 79

Figure 43. Comparison of velocity vectors in Region 5: Left is upstream and right is downstream portion of the region (GSTAR-W is the former name of SRH-2D) 80

Figure 44. Comparison of velocity vectors in Region 6: Left is upstream and right is downstream portion of the region. (GSTAR-W is the former name of SRH-2D) 81

Figure 45. Comparison of velocity vectors in Region 7 (GSTAR-W is the former name of SRH-2D) 81

(Intentionally blank)

SUMMARY

SRH-2D, Sedimentation and River Hydraulics – Two-Dimensional model, is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems under development at the Bureau of Reclamation. It was evolved from SRH-W which had the additional capability of watershed runoff modeling. Version 2, SRH-2D v2, focuses specifically on 2D modeling of river systems for flow hydraulics and geomorphic assessment, with many features improved from SRH-W. Future versions will add additional modules related to sediment, temperature and vegetation modeling. This report serves as the theory and user's manual for SRH-2D v2. The manual provides an introduction to SRH-2D, its unique capability, and its potential applications. Mathematical formulation, numerical methods, and solution algorithms are presented; sample calibration and verification cases are simulated and discussed; and a number of project applications are reported. The manual also provides training to prepare users to simulate river flows using SRH-2D. This manual should be sufficient for users to learn how to apply SRH-2D.

SRH-2D solves the 2D dynamic wave equations, i.e., the depth-averaged St. Venant equations. In terms of modeling capabilities, SRH-2D is comparable to many existing models such as RMA-2 (US Army Corps of Engineers 1996) and MIKE21 (DHI software 1996). SRH-2D does possess a few salient features. First, SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. In practice, the hybrid mesh of quadrilateral and triangular cells is recommended though purely quadrilateral or triangular elements may be used. A hybrid mesh may achieve the best compromise between solution accuracy and computing demand. Second, SRH-2D adopts very robust and stable numerical schemes with seamless wetting-drying algorithm. The outcome is that the model is very stable, and few tuning parameters are needed to obtain reliable solutions. The model has been verified, validated, and successfully applied to numerous flow cases.

The first five Chapters are strongly recommended for new users. The rest of the chapters are for references only.

(Intentionally blank)

CHAPTER 1

INTRODUCTION

This document serves as the theory and user's manual for SRH-2D version 2, a version released for flow simulation in rivers. Other modules for sediment, temperature and vegetation modeling will be released in the future. For watershed runoff modeling, users should use SRH-W version 1 that is downloadable from the same website.

The manual is organized as follows: the background and model capabilities are discussed in this chapter; SRH-2D modeling procedure, model setup commands and tutorials are in chapter 2 through 5; details of the mathematical formulation, numerical methods and solution algorithms are presented in chapters 6 through 8; and sample verification cases are discussed and a number of practical application cases are reported in chapter 9 and 10. The manual may be used to train a new user to understand and use SRH-2D for modeling through self-learning and within a short time.

1.1 General

SRH-2D, Sedimentation and River Hydraulics – Two-Dimensional model, is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems under development at the Bureau of Reclamation. Version 2 focuses specifically on river system modeling for flow hydraulics. SRH-2D v2 is an improvement from its predecessor SRH-W. However, SRH-W has the additional watershed runoff modeling capability and is still available on the same website. Future versions will add additional modules related to sediment, temperature and vegetation modeling.

SRH-2D v2 may be applied but not limited to:

- Flow in one or multiple streams covering the main channel, side channels, and floodplains;
- Flood routing and inundation mapping over any terrain;
- Flow around in-stream structures such as weirs, diversion dams, release gates, coffer dams, etc.;
- Flow over-spill over banks and levees;
- Flow over vegetated areas and interaction with main channel flows;
- Flow in reservoirs with known flow release; and
- Morphological assessment of bed erosion potential.

A number of papers may be referred to for additional information related to modeling and application issues. Theory was presented in Lai (2009; 2010), and some early applications were reported in Lai (2005) and Bountry et al. (2006). Selected project applications, with reports, are available at the SRH-2D website. Additional papers are also available at the website.

SRH-2D was developed with a vision to provide reliable solutions with reasonable turnaround time on a Personal Computer (PC). Advanced solution algorithms are adopted so that it might provide solutions with little parameter turning. SRH-2D is also developed with the objective that a 2D model does not have to be too complex to use. With SRH-2D, users do not have to memorize many commands; they are guided by a preprocessor, an interactive user interface, through the Partial-Interface mode discussed later. Most user input errors may be automatically detected by the preprocessor so errors may be removed before carrying out the final analysis.

1.2 Modeling Concept and Capabilities

SRH-2D adopts a zonal approach for coupled modeling of main and side channels and floodplains. A river system is divided into modeling zones and each zone may be assigned with different parameters such as the Manning's roughness coefficient and may be meshed differently. The zonal partition, along with the mesh layout, is illustrated in Figure 1. A river system is represented by a solution domain. The domain is then partitioned into zones (polygons). A zone may represent an arbitrary flow area. Typically, zones are delineated based on natural features such as the topography, vegetation, and bed roughness.

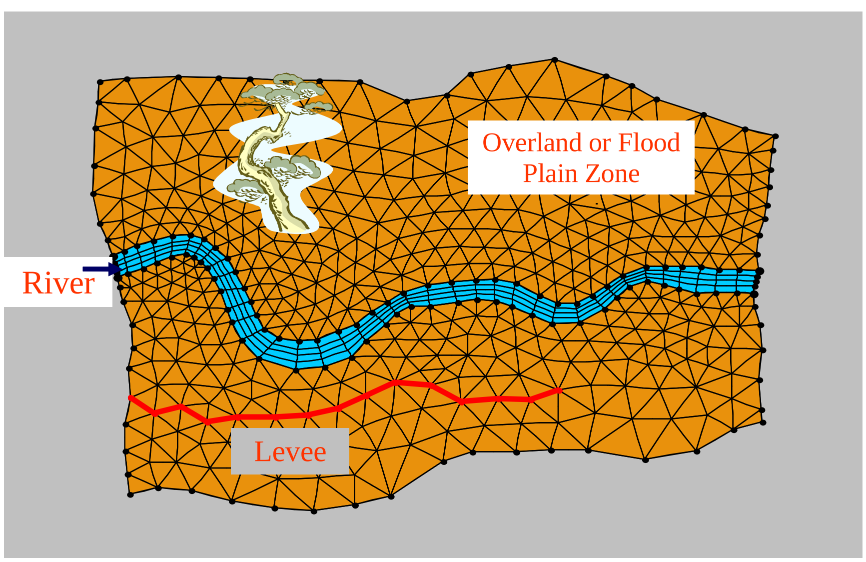


Figure 1. Illustration of zonal partition and mesh layout

One of the salient features of SRH-2D is the use of the hybrid mesh, which is based on the arbitrarily shaped element method of Lai (1997, 2000) for geometry representation. This unstructured hybrid meshing strategy is flexible that facilitates the implementation of the zonal modeling concept. SRH-2D essentially allows the use of most existing meshing methods available, such as the structured curvilinear mesh (pure quadrilaterals), conventional finite element mesh (purely triangles), Cartesian mesh (purely rectangular or square mesh), and the hybrid mixed element mesh. Typical meshes used by SRH-2D are the hybrid mesh as shown in Figure 2 and Figure 3. More mesh examples may be found in Chapter 10.

Figure 2. A sample SRH-2D mesh: quadrilateral cells are used along the main channel and levees but mixed coarser cells are in the floodplains

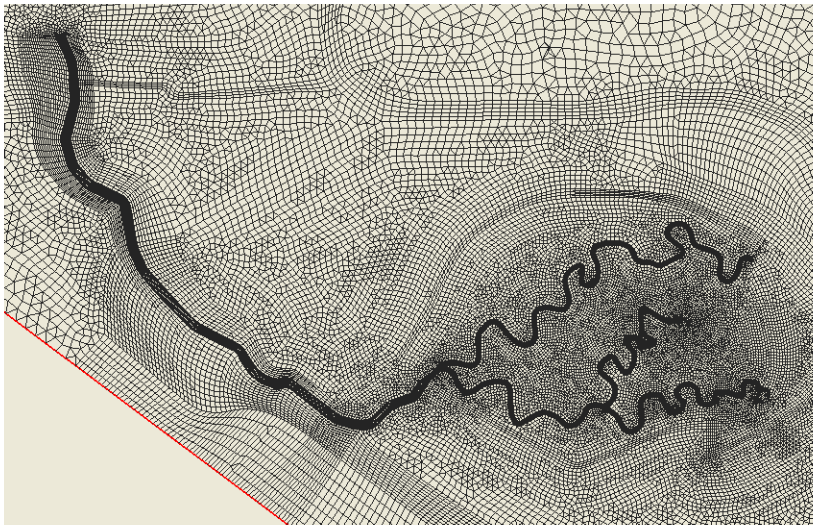


Figure 3. A sample SRH-2D mesh that uses a combination of structured quadrilateral cells and unstructured mixed-shape cells

Major SRH-2D capabilities are listed below:

- 2D depth-averaged dynamic wave equations (the standard St. Venant equations) are solved with the finite-volume numerical method;
- Steady state (with constant discharge) or unsteady flows (with flow hydrograph) may be simulated;
- An implicit scheme is used for time integration to achieve solution robustness and efficiency;
- An unstructured arbitrarily-shaped mesh is used which includes the structured quadrilateral mesh, the purely triangular mesh, or a combination of the two. Cartesian or raster mesh may also be used. In most applications, a combination of quadrilateral and triangular meshes is the best in terms of efficiency and accuracy;
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need for special treatments;
- Robust and seamless wetting-drying algorithm;
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output flow variables include the above, plus Froude number and bed shear stress; and
- Morphological assessment based on the flow solutions, which provides information about the sediment transport capacity rate, critical bed sediment diameter, and shields parameter.

SRH-2D is a 2D model, and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with side channel and agricultural returns, and with braided channel systems. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns, flow recirculation, lateral velocity variation, and flow over banks and levees.

1.3 Limitations

SRH-2D v2 has the following limitations:

- Only flow is modeled with version 2. Mobile-bed sediment transport and other modules are not available at present;
- Only the flow routing module has been developed and released. Users need to have access to other software for mesh generation and result post-processing. At present, SRH-2D v2 uses SMS, the Surface-Water Modeling System (http://www.aquaveo.com/), as its mesh generator, user interface and post-processing. Other graphical post-processing software may also be used such as ArcGIS and TECPLOT. Details are discussed later in this manual; and last but not the least,
- Only personal computers with the Windows Operating System are supported.

1.4 Acquiring SRH-2D

The latest information about SRH-2D may be found on the Web by accessing http://www.usbr.gov/pmts/sediment and following the SRH-2D link on the left of the web page.

SRH-2D is under continuous development and improvement. Users are encouraged to check the SRH-2D web page for updates.

1.5 Disclaimer

SRH-2D and related information in the manual are developed for use at the Bureau of Reclamation. Despite many successful applications of SRH-2D to projects, Reclamation does not guarantee the performance of the program. Reclamation assumes no responsibility for the correct use of SRH-2D and makes no warranties concerning the accuracy, completeness, reliability, usability, or suitability for any particular purpose of the software or the information contained in this manual. SRH-2D is a program that requires engineering expertise to use and for correct result interpretation. Like other computer programs, SRH-2D is potentially fallible. All results obtained from the use of the program should be carefully examined by an experienced engineer to determine if they are reasonable and accurate. Reclamation will not be liable for any special, collateral, incidental, or consequential damages in connection with the use of the software.

CHAPTER 2
GETTNG STARTED

This chapter provides an overview of the SRH-2D model and what users need to know before using SRH-2D. This chapter is mandatory for new users.

2.1 Model Structure

Three programs are needed for a complete analysis with SRH-2D: (1) a mesh generation program; (2) the SRH-2D package; and (3) a post-processing graphical program.

Each of the three programs is described next.

2.1.1 Mesh Generation Program

SRH-2D does not contain the mesh generation program at present. Instead, SRH-2D relies on a third-party mesh generation program. Any 2D mesh generation program may be used since SRH-2D adopts the arbitrarily-shaped mesh system. In general, a combination of quadrilaterals and triangles is the most common mesh type used by SRH-2D. However, the purely quadrilateral mesh or triangular mesh may also be used. If a user has access to a particular mesh generator and would like it be included into SRH-2D, please contact the SRH-2D developer: Dr. Lai (ylai@usbr.gov).

SMS, the Surface-Water Modeling System, is the mesh generator supported by SRH-2D at present. SMS is a pre- and post-processor for surface water modeling and design, which may be obtained with a reasonably-priced license fee. The following website provides more information: http://www.aquaveo.com/ . Only three SMS modules are needed to run SRH-2D: Map Module, Scatter module, and Mesh Module.

APPENDIX A provides some discussion on how to use SMS to prepare a mesh for SRH-2D. Users, however, should get some training to use SMS directly from Aquaveo, LLC.

SMS may be used in one of two ways in conjunction with SRH-2D: (1) Partial-Interface mode: only mesh and boundaries (nodestrings) are generated within SMS and used as inputs to SRH-2D; or (2) Full-Interface mode: all model inputs are set up within SMS, in addition to mesh and boundaries. Full-Interface mode is discussed in Chapter 3 and Partial-Interface mode is explained in Chapter 4. For beginners, Full-Interface is recommended; experienced users may use Partial-

Interface as it offers more controls on running the program, and may have more capabilities.

2.1.2 SRH-2D Package

The SRH-2D release package consists of two programs: ***srhpre*** and ***srh2d***

Srhpre is a text-based interactive user interface that guides a user to set up the SRH-2D simulation in an easy-to-understand manner. It is primarily used in conjunction with Partial-Interface mode. It may be interpreted as a preprocessor to obtain an input file to run ***srh2d***. The interface is designed such that a user does not need to memorize many input commands. The interface has the error checking capability so that errors may be detected before running SRH-2D program. A mesh file with the "2D Generic Mesh" format, generated by SMS with extension .2DM, should have been ready as the input. The 2DM mesh file contains at least the following information:

- Partial-Interface mode: mesh elements with material type (E4Q and E3T cards), mesh nodes (ND cards), and nodestrings (NS cards); and
- Full-Interface mode: mesh elements with material type, mesh nodes, nodestrings, Manning's roughness coefficients (specified within each material type), SRH-2D parameters, and boundary conditions (types and values).

Srh2d is the main solver that reads the input data generated by ***srhpre***, carries out the simulation, and outputs the simulated results to data files in a format accessible to graphical post-processing packages. The output data files contain the final results and may be viewed and processed using the selected graphic packages such as SMS, TECPLOT, or ArcGIS (see APPENDIX B for discussion).

Among output files, a restart or hot-start file, named _RST*n*.dat (*n* is an integer), is created whenever ***srh2d*** is run. The _RST*n*.dat file contains all model information and may be generated at a specified time interval during program execution. The restart file serves several important purposes:

(1) In the event of a computer crash, the program may be continued from the previously saved restart file so that the simulation is not completely lost.
(2) A user may examine the results at the end of a simulation to monitor the solution progress or check whether a steady-state solution has been achieved. The job may be continued to the next or final stage by restarting from the previous run using the _RST*n*.dat file.
(3) For some cases, solutions from the _RST*n*.dat file of another run, but with the same mesh, may be used as an initial condition to speed up the steady state modeling.

(4) The _RST*n*.dat file of a steady-state solution is often used as the initial condition for an unsteady simulation.
(5) A restart file may be used to output the results to another output format, e.g., from SRHN to TECplot.

The restart file is used by SRH-2D in two ways. In the first, it is used for hot-start, controlled by a parameter named *IREST*, so that previous simulation may be continued to completion without any changes in inputs. The parameter *IREST* is set up using the SRH-2D dynamic input file as explained in APPENDIX C. In the second, a restart file from another solution but with the same mesh is used as an initial condition for a new simulation. For such a use, *IREST* should not be used (set it to zero); the INITIAL_CONDITION parameter should be set up as explained in Chapter 3 for the Full-Interface mode and Chapter 4 for the Partial-Interface mode.

2.1.3 Post-Processing Program

SRH-2D outputs intermediate and final results to result files that may be viewed and examined by post-processing graphical programs. Four formats are currently available: SRHN, SRHC, TECPLOT, and XMDF. SRHN is a data file in the ASCII column format; all output variables are at the mesh nodal points (nodes). SRHC is an output file in the ASCII column format; it is the same as SRHN format except that all variables are at the cell (element) centers. Both SRHN and SRHC files may be imported into SMS, ArcGIS, or EXCEL for graphical display and result processing. SMSC stores the naturally computed variables at the cell centers. The disadvantage, however, is that only the mesh center coordinates are the output although the input mesh provides nodal points, so that "half" of the boundary mesh is lost by this format. TECPLOT format is a special form used for result post-processing by the TECPLOT program. The XMDF format is a special output format used by SMS. The advantage is that only one file, casename_XMDF.h5, is created for unsteady simulations and SMS can handle this format much faster. The SRHN, TECPLOT and XMDF formats store all variables at the mesh points (nodes) through interpolation from cell center to nodes. The output results are consistent with the input mesh points but interpolation process may introduce small errors.

2.2 Modeling Steps

SRH-2D solves all equations in SI units (e.g., distance and mesh coordinates are in meters, elevation and water depth in meters, velocity in m/s, stress in N/m^2, etc). For model input and results output, however, users have the option of using either SI unit or the ENglish unit. The specific unit requirement is clearly indicated during the model setup stage. Units are also appended to the variable names in the result output files.

A typical modeling process consists of four steps: mesh generation, preprocessing, model execution, and result post-processing. They are described below.

2.2.1 Mesh Generation

The first step in using SRH-2D is to prepare a 2D mesh using a mesh generation program. At present, SMS is supported by SRH-2D for mesh generation. Note that only the "2D Generic Mesh" coverage, with 2DM extension, is used by SRH-2D. This model coverage should be set up in SMS and only linear elements (versus quadratic elements) are used. Once the mesh is generated, SRH-2D model parameters may also be setup within SMS under the Full-Interface mode, as described in Chapter 3. Mesh generation is discussed in Appendix A.

A mesh may be generated using any units, such as meter, foot, etc. The mesh unit (or scale) information is one of the input parameters used by SRH-2D for conversion.

2.2.2 Preprocessing

Once a mesh is generated in SMS, ***srhpre*** may be started by clicking the executable. A window will pop up that allows an interactive session to begin. The window may be resized to fit the monitor size. A user will be asked to choose the preprocessing mode: Partial-Interface or Full-Interface. Some users had trouble using the Full-Interface mode due to older SMS versions used. The Partial-Interface mode is recommended if difficulty appears. Also, APPENDIX E is recommended before running SRH-2D to avoid some common errors.

If Full-Interface mode is chosen, the SMS 2DM mesh file, say case.2DM, is the only input file. The ***srhpre*** will simply ask the input of the 2DM mesh file name. No more inputs are needed as all SRH-2D input parameters are set up within SMS. Detailed discussion is provided in Chapter 3.

The use of the 2DM mesh file is important:

- It allows a user to keep a permanent record of case simulated. With casename.2DM file saved, a simulation may be repeated later if necessary to reproduce the model results.
- An experienced user may edit casename.2DM directly for simulation setup. This is particularly useful when only minor changes are needed for carrying out a parametric study, or input errors are to be corrected. See Chapter 3.

If the Partial-Interface is chosen, ***srhpre*** allows a user to set up the simulation through an interactive menu-driven session, i.e., commands are entered one by one as directed by the preprocessor. In the beginning, a case name is needed so that all I/O files may use this case name as the identifier. In this manual, "case" is the case name for convenience of discussion unless otherwise stated. During the interactive preprocessing, all inputs are saved to a 'script' file named case_SOF.dat (Script Output File). This file may later be used to rerun ***srhpre*** by renaming case_SOF.dat to case_SIF.dat (Script Input File). The _SIF.dat file may be interpreted as the input file to run the preprocessor. The importance of _SIF.dat file is described below:

- A user may stop ***srhpre*** at any time during the preprocessing step; ***srhpre*** execution may then be continued later from the stop point of last preprocessing using the _SIF file. It is similar to the restart or hot-start procedure of the ***srh2d,*** and it may be necessary so that a user may take a break, or correct input errors.
- It allows a user to keep a permanent record of the simulation once completed. With _SIF.dat file saved, along with the mesh file, a simulation may be repeated later if necessary to reproduce the model results.
- An experienced user may edit the _SIF.dat file directly for simulation setup. This is particularly useful when only minor changes are needed for carrying out a parametric study, or input errors are to be corrected. Therefore, it is recommended that the script file case_SOF.dat be saved to case_SIF.dat immediately after it becomes available, as the _SOF.dat file will be over-written if ***srhpre*** is executed again.

The case_SIF.dat is an ASCII file; each line is designated as either a COMMENT line or a COMMAND line. A comment line starts with '//'. A user may add lines of comments to the script file to assist the interpretation of the input file. The command line is the actual input text which is read and processed by ***srhpre***. A user is encouraged to do an exercise: run ***srhpre*** first with a sample session using the on-screen interactive option, and then examine case_SOF.dat to learn the script file format.

A list of all input commands used by the Partial-Interface mode is discussed in Chapter 4.

After a successful ***srhpre*** session, a data file, case.dat, is created which serves as the input file to the main solver ***srh2d***. Sample execution of SRH-2D is presented in Chapter 5: Tutorial.

2.2.3 Main Solver Execution

A user may start executing the main solver by clicking ***srh2d20*** in a PC window. A number of windows will pop up providing model solution progress and result monitoring. Detailed discussion may be found in Chapters 5.

2.3 SRH-2D Output Files

During and after the execution of SRH-2D, a number of output files are generated and important ones are described below:

2.3.1 _RES File

A file, named case_RES.dat, is created by SRH-2D; this is the solution residual file that records the history of the solution process. For each time step, residuals of each governing equation, normalized to order one, are recorded in the _RES file. Users may check the _RES file directly to monitor the solution progress. For example, it provides information on the status of solution convergence and/or divergence. For a steady state simulation, the solution is probably diverging if residual keeps increasing. Residuals are difficult to define and sometimes it is impossible for them to drop to a low level. This mostly happens at a few locations, at wetting/drying cells, but the overall solution may have already been converged. Therefore, a better indicator for convergence is to check the flow rate through a monitor line near the exit. Results at monitor points may also be used as discussed later.

2.3.2 _OUT File

The _OUT.dat file is an informational file, named case_OUT.dat, which records some basic messages about the simulation run, such as the cpu time, problem definition, etc.

2.3.3 _RST*n* File

The _RST*n*.dat file is the restart or hot-start file, named case_RST*n*.dat, that may be used to continue the simulation from a previous execution. Details have been discussed in Section 2.1.2.

2.3.4 Result Output File

Result output files are used for graphical post-processing. Several formats may be used as discussed in APPENDIX B. Depending on the format selected, case_SRHN*n*.dat is generated if SRHN format is used, case_TEC*n*.dat file is created if TECPLOT format is selected, case_SRHC*n*.dat is generated if SRHC format is chosen, and case_XMDF.h5 is generated for the XMDF format.

CHAPTER 3

FULL-INTERFACE MODE WITH SMS

This chapter provides instructions on how to use SMS as a full interface to SRH-2D, not just as a mesh generator. With Full-Interface mode, the use of preprocessor, ***srhpre***, is limited to reading the SMS 2DM mesh file and checking potential errors. It is not the intent of this Chapter to train users to use SMS for mesh generation; for such a purpose users should consult the SMS user's manual and resort to SMS training classes. This Chapter focuses on how to interface between SMS and SRH-2D. A general guide on mesh generation using SMS is in Appendix A. The SMS modules used by SRH-2D are the Map, Mesh, and Scatter modules.

Caution: some users ran into various troubles using the Full-Interface mode due to older SMS versions used. If such difficulties occur, Partial-Interface mode is recommended. Also, APPENDIX E should be referred to before running SRH-2D to avoid some common errors.

3.1 SRH-2D Template File

A SRH-2D template file is supplied with the release package so that SMS may be configured as the Full-Interface mode to run SRH-2D. It may be done permanently in SMS, or only done for an individual case. The template file named srh2d-sms-template-v10.2DM is for users who use SMS version 10 and above. Another template file, srh2d-sms-template-v8.2DM, is also supplied for users who use SMS version 9 and older. If a user runs into trouble with version 10 (v10) template file, version 8 (v8) template file may be used even if your SMS is version 10 and higher. However, the template file is not needed for Partial-Interface mode.

<u>Procedure to configure SMS as SRH-2D Full-Interface mode permanently</u>: A once-for-all configuration may be done to set up SMS as the Full-Interface for SRH-2D with version 10 and higher. After completion, SMS is automatically has SRH-2D as its default model every time it is started. The procedure is as follows:

- Create a Short Cut for the SMS on the desktop if a user does not already have;
- Right click SMS executable short cut; and choose "properties" by left clicking on the pop-up window (see Figure 4 for the pop-up window);
- Select "Shortcut" button; the content of "Target" already points to the SMS executable such as "C:\Program Files\SMS 10.0\sms100.exe"

- Append now the directory tree of the template file to the end of the "Target". For example, if srh2d-sms-template-v8.2DM is placed in "C:\Program Files\SMS 10.0\" (the same location of sms100.exe), the modified "Target" content would be: "C:\Program Files\SMS 10.0 \sms100.exe" "C:\Program Files\SMS 10.0\srh2d-sms-template-v8.2DM"
- Click "Apply" and it is done (See Figure 4 for the final look).

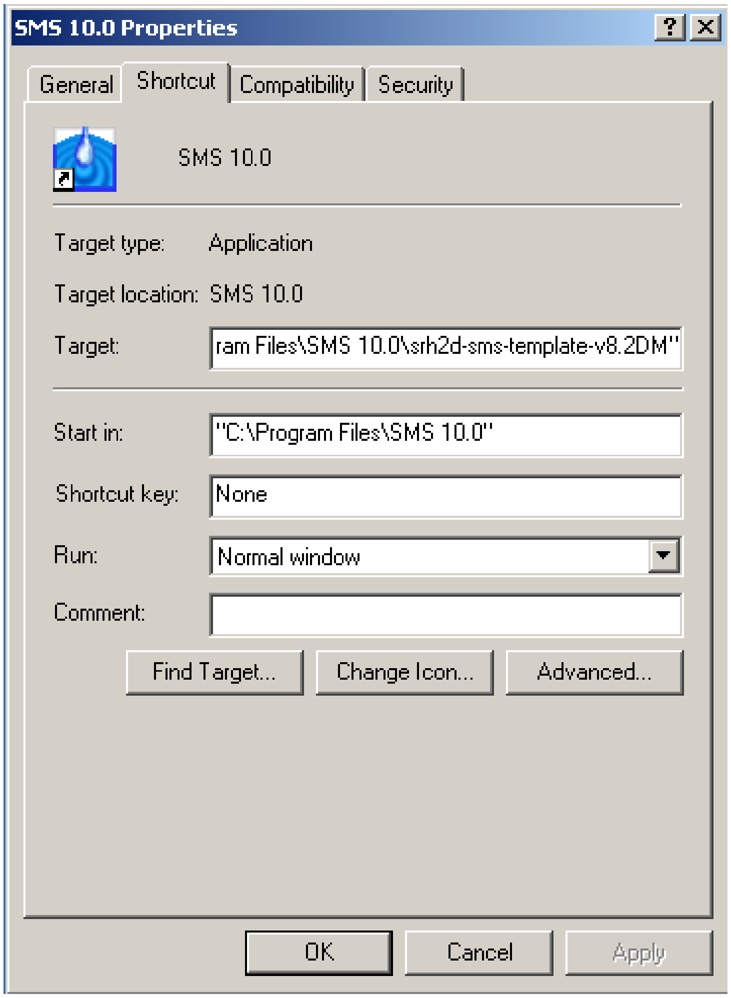


Figure 4. The pop-up window to append the template file to SMS executable

If the SMS shortcut is clicked, the SMS start-up window would now have SRH-2D set up as the default model with the Full-Interface mode. "SRH-2D" should be displayed on the top right corner of the window, next to "Elements". Choose "U.S. Survey" if prompted and click "OK" if a complaining message shows up as they are irrelevant to SRH-2D.

Setup SMS as SRH-2D Full-Interface temporarily: Users also have the option to setup SMS to work as Full-Interface mode for SRH-2D for a specific model run only. It is done by loading the template file into SMS once for each model case, either before or after a mesh is generated. Load the template file into SMS through "File\Open" option. The template file should only be "Appended" to the existing mesh if a mesh already exists. With the v8 template, "Switch Current Model …" under Data button should be done first to set the model as "Generic" before loading the template.

Three groups of SRH-2D input parameters need to be assigned with the Full-Interface mode; they are done after a 2D mesh has been generated within the mesh module. The three groups are: Global Parameters, Boundary Conditions, and Manning's Coefficient. These input parameters may be examined or "edited" directly within the 2DM mesh file. They are located near the last section of the file and each parameter has a "record name" and the associated value. They will be described next.

3.1.1 Global Parameters Setup

After a 2D mesh is generated, the option of "**SRH-2D\Global Parameters …**" becomes available for setup in the mesh module under SRH-2D. These are the global input parameters for SRH-2D. With the v10 template, input parameters are divided into four groups: Global, Flow, Output, and Data_File_List. With the v8 template, no grouping is used.

Steady or Unsteady Solution:

Entering **SRH-2D\Global Parameters …**, users are given the opportunity to select "Steady State" or "Dynamic" under the "General" button (group). "Steady State" should be selected if a steady state simulation is to be carried out with a constant flow discharge; "Dynamic" is chosen for an unsteady simulation when a flow hydrograph is supplied for discharge. "Time step" is in second and is used for SRH-2D time integration; "Total time" is the total simulation time in hours within SRH-2D. Both time step and total simulation time may also be set up with the _DIP.dat file as explained in Appendix C, and is the recommended way of their inputs. It appears that users may not be able to set up time step and total simulation time if "Steady State" is chosen with some SMS versions. One may get around this by clicking "Dynamic" first, setting up time step and total simulation time, and then returning to "Steady State". Users are encouraged to use the _DIP.dat file to set up dt and simulation time (dtnew and Total_Simulation_Time).

"DY" record contains the above setup in the 2DM file: 0 for steady state and 1 for unsteady. "TD" record contains dt and total simulation time.

Simulation_Description

It provides users with the opportunity to describe the kind of simulation to be carried out. A text string is expected. *PD "Simulation_Description"* with the "PD" record is used in the 2DM file and it belongs to the "Global" group.

Case_Name

One word is used to define the case name of the simulation. For convenience of discussion, *case* is assumed as the case name throughout this manual. Users may use any word for the case name, but space, comma, and a few other special characters cannot be used. The case name is used to identify all input and output files. For example, the input file created by ***srhpre*** is named *case*.dat, the result output file is named *case*_SMSn.dat, restart file is named *case*_RSTn.dat, and so forth. A text input is expected. *PD "Case_Name"* with the "PD" record is used in the 2DM file and it belongs to the "Global" group.

Mesh_Unit

The mesh may be generated with a number of unit systems for the horizontal and vertical coordinates. Six options are available: FEET, METERS, MILES, INCHES, MILLIMETERS, and KILLOMETERS. One option should be selected with v10 template and a text should be typed with v8 template. *PD "Mesh_Unit"* with the "PD" record is used in the 2DM file and it belongs to the "Global" group.

Solution_Module

This input parameter selects the module of SRH-2D to be activated. At present, only FLOW module is used, and MORPH can only be used with the Partial-Interface mode. In the future, sediment, temperature, and vegetation modules will be added. It is under the "Global" group with v10 template file. Note: since only FLOW is available with Full-Interface mode, this option will not appear.

Turbulence_Model

Two turbulence models are available: the parabolic model (Zero-Equation) or the two-equation k-ε model (KE-Equation). The depth-averaged parabolic turbulence model calculates the turbulent viscosity with $\nu_t = \alpha V_* h$, where V_* is the frictional velocity and h is the water depth. Coefficient α ranges from 0.3 to 1.0 and the default value of 0.7 is used by SRH-2D. A user has the option to use a different α with the _DIP.dat file. In general, however, results may not be sensitive to α for most applications. The parabolic model works well for most field applications and requires less computing time and is recommended for use first. Also, it is not recommended to use turbulence model or the viscosity as a primary calibration parameter. Solution stability is not affected by the turbulence model. With v10 template, an option may be selected; but with v8 template a text input, ZERO or KE, is the expected input. *PD "Turbulence_Model"* with the "PD" record is used in the 2DM file and it belongs to the "Flow" group.

Initial_Condition

An initial condition is needed for all simulations. For a steady state simulation, the initial water surface elevation may be important in obtaining convergence. Three options are available: DRY, AUTO, or filename. If the text DRY is the input, the entire solution domain is assumed to be dry initially. Zero velocity and zero water depth are set up everywhere. Water may enter the solution domain from both inlet and exit boundaries. If the water surface elevation is too high at the exit, “huge” flow may enter the domain from the exit and a very small time step may be needed initially for some problems. With DRY option, a longer computing time may be needed for problems with a long river reach, large storages, multiple side channels, or a small flow discharge to attain a steady state solution. The second option, AUTO, is the same as DRY except that (1) initial water surface elevation is set up automatically in the solution domain based on the water elevation at the exit, and (2) water will not enter the exit boundaries. The AUTO option may be used for most applications. With the third option, use a text other than DRY and AUTO, say *case_init*. The initial condition will be from the results contained in a restart (hot start) file named *case_init*_RST.dat. The RST file is typically from another SRH-2D run; but it is required that the same mesh is used for both simulations. One use of the RST option is to utilize a steady-state solution as the initial condition for an unsteady simulation; another usage is to use results at another flow discharge as the initial condition for the new steady state run. Note that the RST option here is different from the restart (hot start) option by setting *irest=1* in the _DIP.dat file. The RST option is only to set up the initial condition for the main dependent variables; but *irest=1* is to set all variables and parameters to continue from a previous simulation to completion. *PD "Initial_Condition(Dry/Auto/Filename)"* with the “PD” record is used in the 2DM file and it belongs to the “Flow” group.

Output_Format

The output results may be written in different formats to files. Four formats are available: SRHN, SRHC, or TECplot. Both SRHN and SRHC are in column-based ASCII format that may be imported into SMS, ArcGIS, or Excel for graphical viewing and result processing. The difference is: SRHC format stores all variables at the mesh element (cell) centers while SRHN stores all variables at the mesh nodal points. TECplot format stores all variables at the mesh nodal points and the file may be imported into the TECPLOT graphical package for post-processing. XMDF is a special format used by SMS. For specifics of these formats, users may refer to APPENDIX B. An option may be selected with the v10 template but a text input, SRHN, SRHC, TEC, or XMDF, is the expected entry with the v8 template. *PD "Output_Formal"* with the “PD” record is used in the 2DM file and it belongs to the “Output” group.

Output_Unit

Two unit systems are available to output the simulation results: the English unit system (feet for elevation and depth, feet/second for velocity, lb/ft^2 for shear stress, ect.) or the International unit system (meter for elevation and depth, m/s for velocity, N/m^2 for shear stress, ect.). The unit of each output variables is pended to the variable name in the output file. An option may be selected with the v10 template while a text input, EN or SI, is the expected input. *PD "Output_Unit"* with the "PD" record is used in the 2DM file and it belongs to the "Output" group.

Data_File_01 through Data_File_10:

Users may specify up to 10 data file names that may be used to provide time series data and rating curve data. These data files are used for boundary condition setup, normally for unsteady simulation, as described in Section 3.1.2 below. New file names should be entered consecutively to replace the names already in the template, e.g., *case_tsf_i*.dat. An unmodified default file name will be ignored by SRH-2D. Each data file, named above, will be read, processed, and stored as either a "time series function" or a "general function" (see Appendix D). They are to be used as boundary conditions at inlets and exits for an unsteady simulation, as explained in the boundary condition section below. Sample usage of the time series data includes: the flow hydrograph at an inlet and the water surface elevation versus time data at an exit. A general function may be a rating curve with the data of discharge versus stage at an exit. Caution: SPECIAL format is required on how these data should be stored in the data file, and they are explained in Appendix D. The format should be strictly followed.

3.1.2 Boundary Condition Setup

Boundaries are defined first by creating "nodestring" in SMS. Normally, only inlets and exits need to be "nodestring-ed" since SRH-2D automatically sets up all exterior boundaries as no-slip walls. Each boundary (nodestring), after created, is then assigned a boundary type and conditions with the "**Assign BC …**" option under the SRH-2D button in the mesh module (or left-click on the nodestring). The procedure is as follows:

1. Create a nodestring where boundary conditions are to be specified. Note that all exterior boundaries have been automatically set up as no-slip walls by SRH-2D. Therefore, only those boundaries which are not walls need to be done. Normally, only inlets and exits need to be set up. Caution: only exterior nodes may be included in the nodestring for all regular boundaries; one may ensure this with the "Control" key, instead of the "Shift" key, in SMS to create the nodestring.

2. Monitor lines are created with interior nodetsrings. Users may create up to 99 monitor lines. Flow discharge through each monitor line is recorded in the output file _LN*n*.dat as a function of timem along with the average water surface elevation. At least one monitor line is recommended near a major exit, recommended as the first monitor line, as the flow discharge at this location may serve as the best indicator that a steady state solution is achieved. The output file name is *case*_LN*n*.dat with *n* the monitor line ID.
3. Left-click to select a nodestring, exterior or interior, and the boundary conditions may be set up by selecting "Assign BC …" option (in the SRH-2D button or with right-click).
4. Monitor points are created by selecting a node and invoke 'Assign BC …" option (in the SRH-2D button or with right-click). Check the "Nodal BC' in the display option of SMS to view the monitor points. Up to 99 monitor points may be assigned. If no monitor points are selected, SRH-2D will issue a warning message and mesh cell number 1 is chosen. At the monitor point, simulated results are recorded as a function of time in the output file named *case*_PT*i*.dat (*i* is the i-th point). Results at monitor points may be plotted to show how variables are changing with time. The water surface elevation at the monitor point number 1 is displayed in the SRH-2D monitoring window.

Below is a description of all available boundary types and the boundary values that may be specified.

INLET-Q: It is an upstream inlet boundary with a subcritical flow. A flow discharge is specified along with the unit and lateral distribution. For a steady state simulation, a real positive value is specified as the constant flow discharge. For an unsteady flow, a negative integer, -*n*, is usually the input though a constant discharge (but positive) may be used. The negative sign indicates a time series file is the input. The integer, *n*, is the time series function ID specified in the "DATA_FILE_0*n*" in the Global Parameter setup. The unit of discharge in the data file is also specified: 1 for cfs and 2 for cms. Time is always in hour. The lateral distribution of the velocity on the boundary may be specified with three options: 1 for Constant Velocity, 2 for constant unit discharge, and 3 for conveyance approach, and there are described in Chapter 7.

EXIT-H: It is a downstream exit boundary with a subcritical flow. Water surface elevation should be specified at this boundary along with the unit. For a steady state simulation, a real positive value is the input as the constant water surface elevation (therefore, elevation below zero is not allowed in SRH-2D). For an unsteady flow, a negative integer, -*n*, is the input. The integer, *n*, refers to the data function specified in the "DATA_FILE_0*n*" in the Global Parameter setup. If it is a time series function, time (hour) versus elevation (ft or meter) is the input data. If it is a general function, discharge(cfs or cms) versus elevation(ft or meter), the rating curve, is the input data. The unit of the data is also specified: 1 for feet and cfs, and 2 for meters and cms.

EXIT-Q: It is a downstream exit boundary with a known discharge where the water flows out of the domain. The input is the same as INLET-Q: discharge, unit, and lateral distribution. Caution: at least one EXIT-H is needed for modeling and EXIT-Q is intended only for cases with multiple exits. It is preferable that EXIT-H is the main exit and EXIT-Q is only a secondary exit with a small flow discharge.

INLET-SC: It is an upstream inlet boundary with a supercritical flow. Both discharge and water surface elevation is specified at the boundary. For a steady state simulation, two real positive values are specified: discharge and water elevation. For an unsteady flow modeling, a negative integer, -*n*, may be the input for either discharge or elevation or both, though constant values may also be used. The integer, *n*, refers to the time series function or the general function (rating curve) specified in the "DATA_FILE_0*n*" in the Global Parameter setup. The unit is also specified for the discharge and elevation: 1 for cfs and feet and 2 for cms and meters. Lateral velocity distribution may also be specified as INLET-Q.

EXIT-EX: It is a downstream exit boundary where the flow is supercritical. No boundary conditions are needed at such exits, neither discharge nor water surface elevation.

WALL: It is a solid wall boundary on which velocity is zero. It may also be interpreted simply as the no-slip boundary. Wall is usually used for river banks and at domain edge whether wet or dry. No boundary conditions are needed at wall boundaries. By default, all exterior boundaries are set up as walls. Therefore, there is usually no need to set up WALL boundaries again.

SYMMETRY: It is a symmetry boundary, and it may be interpreted as a slip wall boundary. No boundary conditions are needed at symmetry boundaries.

MONITOR LINE: It is not a real boundary at all; it is an internal polyline which may be used to monitor the total flow discharge through it. At least one monitor line near a major exit is recommended as the discharge through the line is often the best indicator that a steady state solution is attained. Note that a monitor line can not be placed on the exterior boundary. If the internal monitor line is setup mistakenly as one of the boundary types above, an error message will be issued by SRH-2D.

MONITOR POINT: Monitor points are created by selecting a node and invoke 'Assign BC …" option (in the SRH-2D button or with right-click). Up to 9 monitor points may be assigned. If no monitor points are selected, SRH-2D will issue a warning message and mesh cell number 1 is chosen. At a monitor point, simulated results are recorded as a function of time in the output file named *case*_PT*i*.dat (*i* is the i-th point). Results at monitor points may be plotted to show how variables are changing with time. The water surface elevation at the monitor point number 1 is displayed in the SRH-2D monitoring window. No extra information is needed for a monitor point.

3.1.3 Manning's Roughness Coefficient

The third and final setup is the distribution of the Manning's roughness coefficient. The option of "**SRH-2D\Material Properties …**" in the mesh module is used for the purpose. A constant roughness value may be assigned to each material created during mesh generation. If users forget to set up Manning's roughness coefficient, zero roughness is set up by SMS and SRH-2D would issue a warning.

CHAPTER 4

PARTIAL-INTERFACE MODE: INPUT COMMANDS

With Partial-Interface mode, SMS is used only for generating a "2D Generic" mesh, with 2DM extension, along with the "nodestring" for boundary conditions. The remaining SRH-2D solution parameters are set up with the SRH-2D preprocessor. Of course, the 2DM mesh file generated for the Full-Interface mode can also be used for the Partial-Interface mode as the mesh and boundary (nodestring) information is contained in the Full-Interface mode 2DM file.

The Partial-Interface mode is recommended if users encounter difficulties in using the Full-Interface mode as such problems have been reported. We cannot change the way SMS operates or the particular SMS version used. Actually, the Partial-Interface mode gives users more error checking opportunities, and is equally easy to use. Also, Partial-Interface may have more modeling capabilities than the Full-Interface mode, such as the MORPH module, etc.

This chapter lists all input commands used by ***srhpre*** if Partial-Interface mode is chosen. It serves as a complete reference of SRH-2D input commands. Description of input commands is listed in the order of their appearances in ***srhpre***. Note that not all commands will appear during an actual session as only relevant commands would appear. Recall that the *case*_SOF.dat file would be created while running ***srhpre***. Users have the option to terminate ***srhpre*** at any time, rename the file to *case*_SIF.dat, and re-run ***srhpre*** from last stop with SIF file.

In the discussion below, some input parameters are mandatory while others are optional. Optional input parameters are put in brackets, e.g., [*PARA*]; default values are assigned if optional parameters are not given.

APPENDIX E is recommended before running SRH-2D to avoid some common errors.

INPUT METHOD SELECTION

Upon starting ***srhpre,*** users are prompted to choose the "Input Method Selection". Two options are available: enter integer 1 if the interactive input is used. This is a must if a user does not have the _SIF.dat input file; enter integer 2 if users already have the script input file, *case*_SIF.dat. Note that ***srhpre*** will always create a new script output file named, *case*_SOF.dat. It is recommended that the script output file be renamed to *case*_SIF.dat after completion of ***srhpre*** so that it may be used as the script input file for future runs.

CASE NAME

One word is used to define the simulation case. For convenience of discussion, *case* is assumed as the case name and this convention is adopted throughout this manual. Users may use any word for the case name. Once entered, case name is used to identify all input and output files for the simulation by SRH-2D. For example, the script input file should be named *case*_SIF.dat, the script output file is *case*_SOF.dat, the input file created by the preprocessor ***srhpre*** is named *case*.dat, and so forth.

SIMULATION DESCRIPTION

This provides users with an opportunity to describe the kind of simulation users are carrying out. Description is limited to one line currently; *CARRIAGE-RETURN (Enter)* key may be entered if users do not want to have any description. The description will appear in the *case*_SOF.dat file.

SOLVEZR SELECTION

Select the solver module for modeling. Two options are available: FLOW or MORPH. FLOW is selected for hydraulic flow modeling; and MORPH is selected if morphological analysis is also carried out in addition to the FLOW module. If MORPH is chosen, more MORPH-specific input parameters are needed and four more output variables are added in the result file.

MORPH related input parameters include the following:

- GENERAL-SEDIMENT-PARAMETERS
  Two parameters are the input: Specific gravity of the sediment (2.65 is the default), and the number of size classes (sed_nclass) for sediment on the bed. One size class, representing the medium diameter on bed, is often used for morphological assessment. But more size classes may be used if bed gradation data is available.

- SEDIMENT-DIAMETER-FOR-EACH-SIZE-CLASS
  Both the lower (d_low) and upper bound (d_upp) of the sediment diameter for each size class is defined. The medium diameter, d_m = SQRT(d_low*d_upp), is computed and used by the model.

- SEDIMENT-TRANSPORT-CAPACITY-EQUATION

A sediment transport capacity equation is chosen to compute the capacity rate. Four options are available: Engelund-Hansen (1), Parker(1990) (2), Wilcox-Crowe (2003) (3), and Meyer-Peter-Muller as modified by Wong-Parker (2006) (4). If Parker(1990) equation is selected, two more parameters are needed: t_ref50 and ee.

- SELECT-REFERENCE-SEDIMENT-DIAMETER
  Select a method to compute the reference sediment diameter (d_ref) which is used to compute the Shields parameter and critical velocity for incipient motion. Three options are available: (1) a constant d_ref may be given by a user; (2) d50 is used as d_ref based on the bed gradation input; and (3) d_nn is used as d_ref based on the bed gradation input (nn represents percentage of sediments that are smaller than d_nn on the bed; e.g., nn=50 gives d50).

- BED-SEDIMENT-GRADATION-INPUT-METHOD
  Bed gradation should be the input and three options are available: (1) "UNIFORM" assumes that the same gradation is used over the entire solution domain; (2) "2DM" indicates that the solution domain is divided into material zones in SMS and each zone has a different gradation. A 2DM file is used to delineate the zone the same way as the material zones used for different Manning's coefficient; and (3) "POINT" is a way to input bed gradations at a number of points, stored in a file, within the solution domain. Gradations on these points are then interpolated to the entire domain.

- BED-SEDIMENT-COMPOSITION
  If "UNIFORM" or "2DM" option is used in the above, the bed gradation (composition) is the input for each zone using one of the two options:

  FRACTION v_1 v_2 ... v_sed_nclass

  or

  CUMULATIVE d1 P1 d2 P2 ... dn Pn

  FRACTION option specifies the gradation with a list of volume fraction (v_i) for each size. And CUMULATIVE option specifies the bed gradation using the cumulative distribution. A list of data pairs, a total of n pairs, are used, and each pair gives (di Pi) with di the diameter in mm and Pi the percentage of sediments finer.

MORPH module will output four additional variables in the results files, in addition to all variables of the FLOW module:

- Sediment Transport Rate (CAP_Q)
  Sediment transport rate, mass per unit width per time, is computed based on the sediment transport equation selected.

- Critical Sediment Diameter (d50_cri)
  Any sediment smaller than d50_cri in diameter are mobilized. Sediment larger than d50_cri will not be moved. The critical diameter d50_cri is computed assuming the critical Shield parameter is 0.04. If other critical Shields parameter is used, d50_cri may be computed based on the fact that Shields parameter is inversely proportional to d50_cri.

- Shields Parameter (Shields_Para)
  Shields parameter is computed as follows:

  $$Shields\_Para = \frac{\tau_b}{\rho_w g(\gamma - 1) d\_ref}$$

  where τ_b is bed shear stress, ρ_w is water density, g is gravity acceleration, and d_ref is the user input reference diameter (see above).

- Mean Sediment Diameter on Bed (d50)
  This is the d50 computed based on the input bed gradation.

RESULT OUTPUT FORMAT AND UNIT

This command specifies the result output file format and units for writing the 2D simulation results to output files. Four formats are currently available: SRHN, SRHC, TECplot, or XMDF. SRHN and SRHC output files are in the ASCII column format for all variables and they may be imported into SMS, ArcGIS, or EXCEL programs. The only difference between SRHN and SRHC is that SRHC stores all data at the mesh cell (element) centers while SRHN stores all data at the mesh nodes. TECplot format is a special form used for result post-processing by the TECPLOT program, and XMDF format is a special format for SMS. Note that all computed variables are located at cell centers and the SRHC format stores the naturally computed variables. The disadvantage, however, is that only the mesh center coordinates are the output although the input mesh provides nodal points, so that “half” of the boundary mesh is lost by this format. The SRHN, TECPLOT, and XMDF formats store all variables at the mesh points (nodes) through second-order interpolation from cell center to nodes. The output results are consistent with the input mesh points but interpolation process may introduce errors.

Two unit systems are available to output the results: SI or EN. Enter SI for SI unit system (e.g., meters for elevation and depth, m/s for velocity, and N/m^2 for shear stress); and enter EN for English unit system (e.g., ft, ft/s, and lb/ft^2). The file name of the result output is dependent on the format chosen; examples are: *case*_SRHN*n*.dat, *case*_SRHC*n.dat* *case*_TEC*n*.dat, or *case*_XMDF.h5 (*n* is a consecutive integer starting from 1).

RESULT OUTPUT AT MONITORING POINTS

This command allows users to specify up to 99 monitoring points where simulated results will be recorded at each time step; that is, time series of output variables are available at monitoring points. X and Y coordinates of each point are given to define the points. The point files are named as *case*_PT*n*.dat with *n* the point number.

The output variable list and the associated variable units at the monitor points may be obtained by examining the headers of the file. Note that only the first monitor point, PT1, is displayed in the monitoring window.

COORDINATES OF MONITORING POINTS

This command specifies the planview coordinates, *X* and *Y*, of all monitoring points. A total of 2*n* real values are needed as inputs where *n* is the total number of monitoring points. The unit of *(X,Y)* should be the same as that of the mesh. If *(X,Y)* point is outside the solution domain, the preprocessor will issue a warning message and this point is ignored. It is recommended that the mesh generation program, such as SMS, be used to determine the coordinates of the monitoring points.

STEADY-OR-UNSTEADY-SIMULATION

SRH-2D runs always in the unsteady mode. If UNSTEADY is chosen, however, results are time accurate meaning all intermediate results are solved correctly and this should be the option for an unsteady flow modeling. With the STEADY option, only the final steady solutions are sought and the intermediate results may not be right. The STEADY option should be selected if only final solutions are sought as it takes much less computing time.

TIME-STEP-AND-SIMULATION-TIME

Three parameters are needed related to time step and simulation time:

TSTART DT T_SIMU

where:

TSTART: a real value for the simulation starting time in HOUR (0.0 is typically used unless there is a good reason otherwise).

DT:	a real value for the time step in SECOND for the simulation.
T_SIMU:	Total simulation time in HOUR to be performed.

Note that both ***DT*** and ***T_SIMU*** may be dynamically changed using the special _DIP.dat file during the SRH-2D execution. See APPENDIX C for more information.

TURBULENCE-MODEL-SELECTION

This command selects the turbulence model to be used. Two models are available: PARA or KE, where PARA=depth-averaged parabolic turbulent model and KE= standard k-ε two-equation model.

The depth-averaged parabolic turbulence model (Rodi, 1993) calculates the turbulent viscosity with $\nu_t = \alpha V_* h$ where V_* is the frictional velocity and h is the water depth. Coefficient α ranges from 0.3 to 1.0, and a default value of 0.7 is used by SRH-2D. Users have the option to use a different α using the _DIP.dat file. In general, final results may not be sensitive to α for most applications and it should not be used as a primary calibration parameter or for promoting numerical stability.

Other turbulent models have been added and only the standard k-ε two-equation model is available. The usefulness of more sophisticated turbulence models other than the parabolic model is yet to be understood. We do not find the turbulence model critical in most applications.

INITIAL-CONDITION-SETUP-METHOD

An initial condition is needed for all simulations. For a steady state simulation, an initial condition for the water surface elevation is important in obtaining convergence or for reduction of computer time. This command allows users to choose one of the several methods to set up the initial condition and they are described below:

- **DRY Bed Setup:** The entire solution domain is dry initially. Zero velocity components and zero water depth are set up everywhere. Water may enter the solution domain from both inlet and exit boundaries. If the water surface elevation is too high at the exit, "huge" flow may enter the domain from the exit and a very small time step may be needed initially for some problems. With DRY option, a longer computing time may be needed for problems with a long river reach, large storages, multiple side channels, or a small flow discharge to attain a steady state solution. This option works for almost all problems.

- **AUTOmatic Setup**: The second option, AUTO, is the same as DRY except that: (1) initial water surface elevation is set up automatically in the solution domain based on the water elevation at the exit, and (2) water will not enter the exit boundaries. The AUTO option may be used for most applications.
- **RST Setup:** The initial condition will be from the results contained in a restart (hot start) file, e.g., *case_init*_RST.dat. The RST file is typically from another SRH-2D run; but it is required that the same mesh is used for both simulations. One use of the RST option is to utilize a steady-state solution as the initial condition for an unsteady simulation; another usage is to use results at another flow discharge as the initial condition for the new steady state run. Note that the RST option here is different from the restart (hot start) option by setting *irest=1* in the _DIP.dat file. The RST option is only to set up the initial condition for the main dependent variables; but *irest=1* is to set all variables and parameters to continue from a previous simulation to completion.

INITIAL-CONDITION: RESTART FILE NAME

This command is to specify the restart file name.

MESH-UNIT

The mesh generated in SMS may be in any unit system. Available options include: FOOT, METER, INCH, MILE, KM, and MM. A text input, one of the listed options, is the expected input and the default is "FOOT".

IMPORT-MESH-FILE

This command specifies the file name and the format of the mesh for importing the mesh into SRH-2D. At present, only one mesh format is used: ***SMS***.

MANNING'S ROUGHNESS INPUT METHOD

This command determines the input method to be used to specify the distribution of Manning's roughness coefficient over the solution domain. Two options are available: (1) enter 1 if a constant Manning's coefficient is used over the entire solution domain; and (2) enter 2 if different Manning's coefficients are specified over different material types of the ***SMS-2DM*** mesh generated using ***SMS***.

CONSTANT MANNING'S COEFFICEINT

This command appears if option 1 is selected in the ***MANNING'S-ROUGHNESS--INPUT-METHOD*** command. A constant Manning's coefficient is provided over the entire solution domain.

NUMBER OF MATERIAL TYPES USED

This command appears only if option 2 is used in the ***MANNING-ROUGHNESS-INPUT-METHOD*** command. The total number of material types is provided to specify the Manning's coefficient.

MANNING COEFFICIENT FOR MATERIAL TYPE

This command appears only if option 2 is used in the ***MANNING-ROUGHNESS-INPUT-METHOD*** command. It specifies the Manning's roughness coefficient for each material type in the ***NUMBER-OF-MATERIAL-TYPE*** command.

SPECIAL-TREATMENT-AT-SELECTED-MESH-CELLS

This option provides some special treatments at user selected zones on the river bed. Some of the special treatments are discussed in Appendix F.

SPECIFY-BOUNDARY-CONDITION-for-NodeString

Each NODESTRING created in SMS is treated as either an exterior boundary segment on which boundary conditions are to be set up or an internal boundary which is set up as a monitor line. The order SRH-2D uses in this command is the same as the order used to create the NODESTRING within SMS. If a user is unsure of which NODESTRING is prompted by SRH-2D, the start and end node IDs printed out by SRH-2D on screen may be used to find the NODESTRING in SMS (node ID may be displayed within SMS).

All exterior boundaries of the solution domain have been automatically set up as a WALL boundary by SRH-2D, a default boundary type. The above default setting may be adequate for most boundary segments. Any segments that are different from a WALL should be included in the NODESTRING lists within SMS and their type and boundary values are set up here with the command. This way, the default WALL setting is overwritten.

The first input required is the boundary type and one of the following is used:

INLET-Q EXIT-H EXIT-Q INLET-SC EXIT-EX WALL SYMM MONITOR

where:

- ***INLET-Q*****:** A subcritical upstream inlet boundary. Flow discharge is to be specified later;
- ***EXIT-H*****:** A subcritical downstream exit boundary. A water surface elevation is to be specified later;

- ***EXIT-Q*****:** An exit through which flow discharge is known. Flow discharge is to be specified on this boundary. Note that this boundary is intended as a secondary exit in addition to the primary exit. At least one primary exit is required with the ***EXIT-H***;
- ***INLET-SC*****:** A supercritical upstream inlet boundary. Both a flow discharge and a water surface elevation are specified at the boundary later;
- ***EXIT-EX*****:** A supercritical exit boundary. Neither discharge nor water elevation is required at this boundary;
- ***WALL*****:** A no-slip solid wall boundary on which velocity is zero;
- ***SYMM*****:** A symmetry boundary. This boundary type may also be regarded as a slip wall boundary; and
- ***MONITOR***: This is not a boundary condition at all. It is an internal boundary through which flow discharge is computed and outputted to *case*_LNi.dat files (*i* is an integer referring to the order of MONITOR lines).

Only the first four boundary types need further boundary value information and they are discussed below.

SPECIFY-BOUNDARY-VALUES

Flow discharge and/or water surface elevation should be further given for the following four boundary types: ***INLET-Q, EXIT-H, EXIT-Q***, and ***INLET-SC***.

They are listed below:

INLET-Q and EXIT-Q need two inputs:	Q UNIT
EXIT-H needs two inputs:	W UNIT
INLET-SC needs three inputs:	Q W UNIT

Q is flow discharge and may take one of two forms: A constant real value or a file name containing a time series hydrograph. A steady state simulation uses the constant discharge; and an unsteady simulation usually uses a time series hydrograph. The hydrograph file should have the following format: (1) the first three rows are comment lines; and (2) starting from the 4^{th} row, two values are given for time (hour) and discharge (Q). The unit of Q is specified by the UNIT option as explained below.

W is the water surface elevation (or stage) at the boundary and may take one of the two forms: a constant value or a file name containing W data. If a constant is given, an unchanging water elevation is assumed. The W file may be a time series data (time~W) or a rating curve. The first row of the file indicates the data type: RATING_CURVE text string on the first row indicates that a rating curve is used; otherwise, a time series is assumed. A time series file has the following format: (1) the first three rows are comment lines; and (2) starting from the 4^{th} row, two values are given for time (hour) and water surface elevation (unit is as discussed below). For a rating curve file: (1) first row has the text RATING_CURVE; (2)

2nd and 3rd rows are comment lines; and (3) starting from the 4th row, two values are given for Q and W (their unit is discussed below). It is cautioned that a rating curve may cause solution instability for some cases. If it occurs, one may convert the rating curve to time series data given a know hydrograph.

UNIT is the unit of Q and W and may be SI or EN. If SI is the input, Q has the unit of m3/s and W has the unit of meters. If EN is given, Q is ft3/s and W is feet.

INTERMEDIATE RESULT OUTPUT CONTROL

This command is to specify the time interval, ***INTERVAL***, in HOUR for intermediate result output. For every ***INTERVAL*** hours, two files will be generated by ***srh2d***: the restart file (*case*_RST*n*.dat) and the result output file (e.g., *case*_SMS*n*.dat or *case*_TEC*n*.dat; *n* is the integer indicating *n*-th output). If ***INTERVAL***<0, no intermediate output will be made and only the final results will be the output.

Intermediate output is recommended as it saves a copy of the restart file (_RST*n*.dat file) so that simulation may be continued in the event of a computer crash. In addition, it offers users an opportunity to examine and view the results to monitor the solution progress.

The parameter ***INTERVAL*** may also be set up and changed dynamically using the _DIP.dat file discussed in APPENDIX C.

CHAPTER 5

TUTORIAL

This chapter provides tutorial cases so that solution processes and procedures may be reviewed with SRH-2D, and inputs and outputs may be familiar to users. The primary purpose is to train users to use SRH-2D with simple examples. The solution process for more complex problems is similar. All tutorial cases come with SRH-2D distribution package, and users are encouraged to run these tutorial cases to get hands-on experience. This chapter is particularly recommended for new users.

5.1 A Subcritical Flow in a Channel

Test case one of MacDonald (1996), a 1D subcritical flow, is used here to serve as a tutorial case to learn how to run SRH-2D. Despite its simplicity, the case covers essential procedures to run SRH-2D; it also trains users getting familiar with the model. More details of the tutorial case may be found later in Section 9.1 1D Subcritical Flow.

Step 1 of the analysis is collection of data relevant to the flow simulated. For the case, these include the solution domain (1000m-by-10m size), bed elevation (provided in analytical form by MacDonald 1996), flow discharge at the inlet boundary (15 m^3/s), water elevation at the exit (0.7484 m), and Manning's roughness coefficient (0.03).

Step 2 involves the generation of a mesh for the solution domain. For the case, a simple 80-by-3 Cartesian mesh cells are generated in SMS as shown in Figure 5. If Full-Interface mode is used, solution parameters, boundary conditions, and Manning's roughness coefficient are also set up in SMS. The 2DM mesh file, named tutorial_c1.2DM, enclosed with the distribution package, is the Full-Interface version output from SMS with the v10 template. It may be used for the tutorial exercise. Users are encouraged to view the 2DM file using a text editor such as Nodepad. It may be seen that the bottom section contains the general parameters, boundary conditions, and the Manning's coefficient. For an experienced user, 2DM file can be modified and edited directly to set up and/or change the parameters.

Users are encouraged to run ***srhpre*** with both Full-Interface and Partial-Interface modes; the same 2DM file can be used since information at the bottom sections containing the general parameters, boundary conditions, and the Manning's coefficient are ignored by SRH-2D. Users are encouraged to try both modes to see whether the same solutions are obtained.

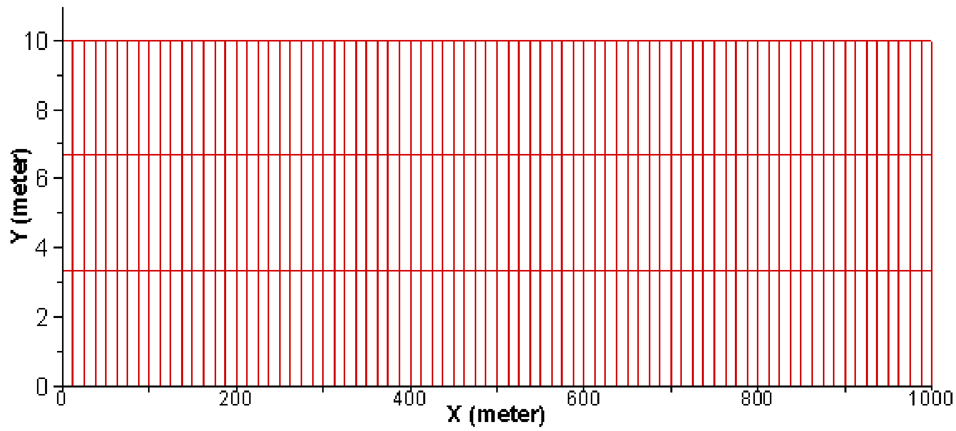


Figure 5. An 80-by-3 mesh used for simulation for test 1 of MacDonald (1996)

Step 3 is to run the preprocessor, ***srhpre***, to develop an input file for SRH-2D solver. Only the 2DM file name, tutorial_c1, needs to be the input for the Full-Interafce mode. With the Partial-Interface mode, however, inputs are guided by ***srhpre***. The case name used in this manual is "c1" and the time step of the simulation is 5 seconds. After completion of ***srhpre***, a script file named c1_SOF.dat is created that may be renamed as c1_SIF.dat. The c1_SIF.dat script file may be used to run ***srhpre*** again with the "Use a Script Input File" preprocessing mode instead of the interactive mode (users are encouraged to try this!). The c1_SIF.dat script file is also included in the distribution package and is not listed here. Once done, a data file, c1.dat, is created which is an ASCII input file for use by ***srh2d***.

Before executing the solver ***srh2d***, a user is encouraged to create the Dynamic Input (DIP) file, c1_DIP.dat, as described in Appendix C. The DIP file used for the tutorial case is included in the distribution package and may be used for other applications (The "a_turb = 0.0" should be deleted though).

Step 4 is to run the solver, ***srh2d***, by simply clicking the executable. Upon entering the case name, c1 for the tutorial case, a window will pop up and several sub-windows will be displayed so that the solution process may be monitored. Figure 6 shows these windows for the tutorial case. One window is the master window (named srh2d) that displays the total cpu time for the simulation. The "Residual Monitoring" window may be used to check the solution process, e.g., the total simulation time (hrs) that has been solved. It also provides residual reduction of the solver (residual may be interpreted as relative error of two velocity equations). Ideally the residual would decrease to a low level. For most natural flows, however, residual will usually stall at a constant value or fluctuate around a mean wildly. This is due to a number of causes. For most cases, it is due to small areas with wetting and drying. For the tutorial case, the residual stalled due to the near zero lateral velocity component of the lateral velocity equation. The individual residual of u- and v-equations may be found in the output file:

c1_RES.dat. Note that the residual is not a good indicator to determine whether a steady state solution has been achieved. Such a check is better done with the flow discharge through a monitor line near the exit. A third window, Water Surface Elevation, displays the calculated water surface elevation at the user-specified monitor point if such a point is supplied. The water elevation at monitor point number 1 is displayed if multiple points may be supplied. A good monitoring point is where the flow is hard to reach. Results at each monitor point are stored in the output file: c1_PT1.dat, c1_PT2.dat, etc.

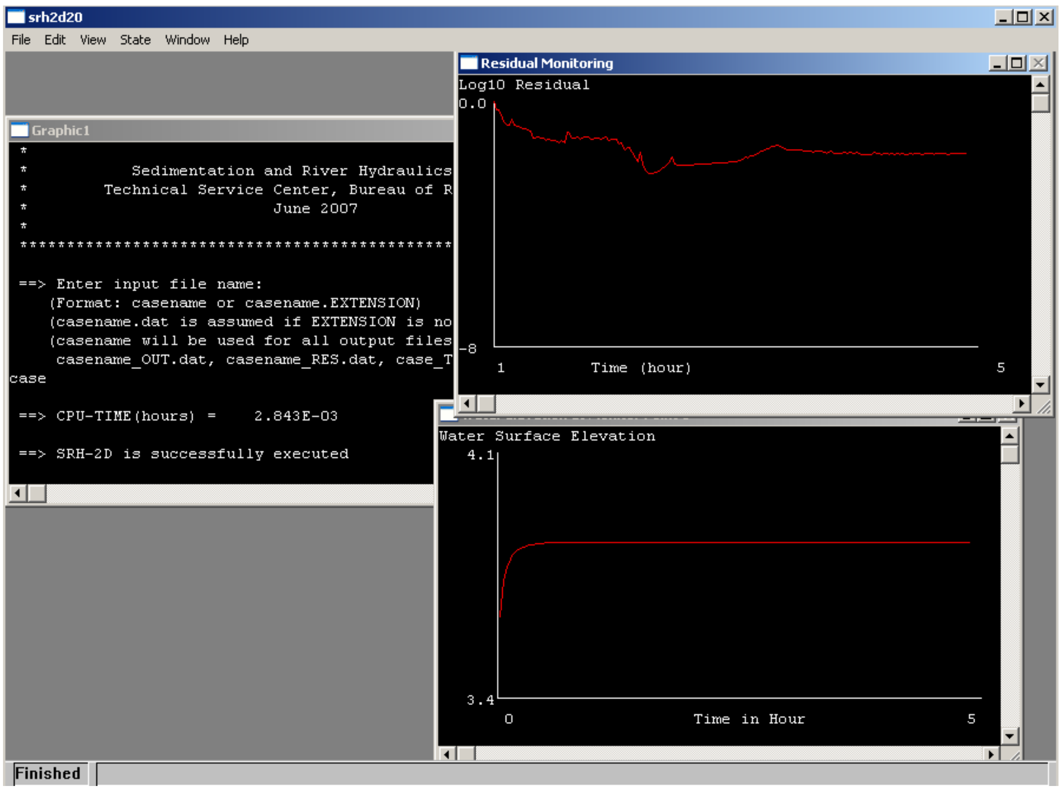


Figure 6. A sample window session running ***srh2d20*** for the tutorial case 1

The final step is to post-process the results. A number of output files are generated after completion of ***srh2d*** execution. They are discussed below for the tutorial case:

c1_DIA.dat: This is the DIAgnostic file with potential errors and warnings about the execution. It helps to identify causes of execution error or failure. For the tutorial case, the file is almost empty indicating a successful run of the model.

c1_OUT.dat: This is the OUTput file providing general model information such as input parameters, mesh size, list of restart file numbers and their corresponding time, cpu time of the simulation, ect.

c1_PT1.dat: It provides time history of output variables at the user-specified monitor points. The file is in column format and may be imported into Excel for plotting. Output from the file may be used to decide if a steady state solution has been obtained or to examine unsteady change of a variable.

c1_RES.dat: It contains residuals of continuity and two velocity equations during the solution. Note that residuals are normalized. For example, the ResH is normalized by the maximum of the first three iterations. Therefore, residual of 1.0 is obtained for ResH if NITER is less than 4 in the c1_DIP.dat file.

c1_RST1.dat: This is the restart file in an unformatted binary form and its intended use has been discussed in CHAPTER 2.

c1_TEC1.dat: It is the result output file with TECPLOT format (see APPENDIX B for more discussion on data format). The output file contains all output variables at the user-specified time and over all mesh points. It may be imported into the corresponding graphical software (TECPLOT for the tutorial case) for viewing and processing of the simulation results.

In general, only the result output file (e.g., c1_TEC1.dat) is important. The restart file is also important if a user intends to continue the simulation from last simulation. A simple way of restarting the run, say up to 10 hours, consists of three steps: (1) set IREST=1 and TOTAL_SIMULATION_TIME=10 in the c1_DIP.dat file; (2) rename c1_RST1.dat to c1_RST.dat; and (3) click the ***srh2d*** to run the model.

For the tutorial case, simulated water surface elevation and water depth are processed using TECPLOT and results are compared with the analytical solution of MacDonald (1996) in Figure 9 and Figure 10.

CHAPTER 6

GOVERNING EQUATIONS

This chapter presents all governing equations used by SRH-2D. It provides theoretical information and is intended for reference only. The paper of Lai (2010) may be referred to for more information.

6.1 Flow Equations

Most open channel flows are relatively shallow and the effect of vertical motions is negligible. As a result, the most general flow equations, the three-dimensional Navier-Stokes equations, may be vertically averaged to obtain a set of depth-averaged two-dimensional equations, leading to the following well known 2D St. Venant equations:

$$\frac{\partial h}{\partial t}+\frac{\partial hU}{\partial x}+\frac{\partial hV}{\partial y}=e \qquad (1)$$

$$\frac{\partial hU}{\partial t}+\frac{\partial hUU}{\partial x}+\frac{\partial hVU}{\partial y}=\frac{\partial hT_{xx}}{\partial x}+\frac{\partial hT_{xy}}{\partial y}-gh\frac{\partial z}{\partial x}-\frac{\tau_{bx}}{\rho}+D_{xx}+D_{xy} \qquad (2)$$

$$\frac{\partial hV}{\partial t}+\frac{\partial hUV}{\partial x}+\frac{\partial hVV}{\partial y}=\frac{\partial hT_{xy}}{\partial x}+\frac{\partial hT_{yy}}{\partial y}-gh\frac{\partial z}{\partial y}-\frac{\tau_{by}}{\rho}+D_{yx}+D_{yy} \qquad (3)$$

In the above, t is time, x and y are horizontal Cartesian coordinates, h is water depth, U and V are depth-averaged velocity components in x and y directions, respectively, e is excess rainfall rate, g is gravitational acceleration, T_{xx}, T_{xy}, and T_{yy} are depth-averaged turbulent stresses, $D_{xx}, D_{xy}, D_{yx}, D_{yy}$ are dispersion terms due to depth averaging, $z = z_b + h$ is water surface elevation, z_b is bed elevation, ρ is water density, and τ_{bx}, τ_{by} are the bed shear stresses (friction). Bed friction is calculated using the Manning's roughness equation as follows:

$$\begin{pmatrix}\tau_{bx}\\ \tau_{by}\end{pmatrix}=\rho C_f\begin{pmatrix}U\\ V\end{pmatrix}\sqrt{U^2+V^2}; \qquad C_f=\frac{gn^2}{h^{1/3}} \qquad (4)$$

where n is the Manning's roughness coefficient.

Turbulence stresses are based on the Boussinesq equations as:

$$T_{xx} = 2(\upsilon + \upsilon_t)\frac{\partial U}{\partial x} - \frac{2}{3}k$$
$$T_{xy} = (\upsilon + \upsilon_t)(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}) \qquad (5)$$
$$T_{yy} = 2(\upsilon + \upsilon_t)\frac{\partial V}{\partial y} - \frac{2}{3}k$$

where υ is kinematic viscosity of water; υ_t is turbulent eddy viscosity; and *k* is turbulent kinetic energy.

A turbulence model is used to compute the turbulent eddy viscosity. Two turbulence models may be used (Rodi 1993): the depth-averaged parabolic model and the two-equation *k*-*ε* model. With the parabolic model, $\upsilon_t = C_t U_* h$ in which U_* is the bed frictional velocity. The model constant C_t ranges from 0.3 to 1.0, and a default value of C_t=0.7 is used by SRH-2D; but its value may be changed using the _DIP.dat file described in APPENDIX C. Note that terms with *k* are dropped in Equation (5).

If *k*-*ε* model is used, turbulent viscosity is calculated with $\upsilon_t = C_\mu k^2 / \varepsilon$. Two additional equations are solved as follows:

$$\frac{\partial hk}{\partial t} + \frac{\partial hUk}{\partial x} + \frac{\partial hVk}{\partial y} = \frac{\partial}{\partial x}(\frac{h\upsilon_t}{\sigma_k}\frac{\partial k}{\partial x}) + \frac{\partial}{\partial y}(\frac{h\upsilon_t}{\sigma_k}\frac{\partial k}{\partial y}) + P_h + P_{kb} - h\varepsilon \qquad (6)$$

$$\frac{\partial h\varepsilon}{\partial t} + \frac{\partial hU\varepsilon}{\partial x} + \frac{\partial hV\varepsilon}{\partial y} = \frac{\partial}{\partial x}(\frac{h\upsilon_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}) + \frac{\partial}{\partial y}(\frac{h\upsilon_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}) + C_{\varepsilon 1}\frac{\varepsilon}{k}P_h + P_{\varepsilon b} - C_{\varepsilon 2}h\frac{\varepsilon^2}{k} \qquad (7)$$

The following definitions and coefficients are used (Rodi 1993):

$$P_h = h\upsilon_t\left[2\left(\frac{\partial U}{\partial x}\right)^2 + 2\left(\frac{\partial V}{\partial y}\right)^2 + \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)^2\right] \qquad (8)$$

$$P_{kb} = C_f^{-1/2}U_*^3; \quad P_{\varepsilon b} = C_{\varepsilon\Gamma}C_{\varepsilon 2}C_\mu^{1/2}C_f^{-3/4}U_*^4 / h \qquad (9)$$

$$C_\mu = 0.09,\ C_{\varepsilon 1} = 1.44,\ C_{\varepsilon 2} = 1.92, \sigma_k = 1, \sigma_\varepsilon = 1.3, C_{\varepsilon\Gamma} = 1.8 \sim 3.6 \qquad (10)$$

The terms P_{kb} and $P_{\varepsilon b}$ are added to account for the generation of turbulent energy and dissipation due to bed friction for the case of uniform flows.

The dispersion terms arise due to the depth averaging process and may become important when secondary flows are present (Flokstra 1976). It was shown by Mihn Duc et al. (1996) that the effect of the secondary flow may be accounted for indirectly by increasing the coefficient of momentum exchange in the horizontal plane.

Some discussion of the Manning's roughness coefficient is in order. With SRH-2D, the Manning's coefficient is a local constant that does not change with flow; but it may be spatially distributed depending on bed types. In addition to the Manning's coefficient, another representation of flow roughness is also convenient with the equivalent roughness height k_s of the bed. For a loose bed, the equivalent roughness height and Manning's coefficient should include both effects of the bed material grain size and bed form. These two parameters may be converted from each other using the Strickler's formula:

$$n = \frac{k_s^{1/6}}{A} \tag{11}$$

where A is in the neighborhood of 26 depending on sediment size, bed form, vegetation, and channel morphology. For flat beds, k_s may take $2d_{90}$, based on the diameter of the bed material. A somewhat higher value, e.g., $3d_{90}$, was used by van Rijn (1987). For sand-wave beds, k_s is related to the wave height.

CHAPTER 7

INITIAL AND BOUNDARY CONDITIONS

SRH-2D needs proper initial and boundary conditions for simulation. This chapter discusses the type of initial and boundary conditions used.

7.1 Initial Conditions

Initial conditions, i.e., values of velocity components (*U* and *V*), water surface elevation (*Z*), and turbulence kinetic energy and dissipation rate (k and ε) if the $k-\varepsilon$ turbulence model is used, are needed to start the SRH-2D simulation. Several ways are offered by SRH-2D to set up the initial conditions.

If only steady state solutions are sought, initial water surface elevation is the only initial variable to be set up, as zero velocity components and small values of k and ε values are automatically set up by SRH-2D. Initial water surface elevation may be set up in several ways including: dry bed setup and restart setup. Readers are referred to Chapter 3 for information for each setup method.

For an unsteady simulation, restart setup is recommended. For example, the initial conditions are from a SRH-2D steady state solution.

7.2 Inlet Boundary

An inlet boundary is defined as a boundary segment on the solution domain where flow is expected to move into the domain. Multiple inlets may be specified for a solution domain. At an inlet, velocity is specified by a user. If sediment transport is also simulated, sediment concentrations at the inlet are also needed.

At present, a total discharge, Q in m^3/s, through an inlet is specified. This discharge may be a constant value for steady state simulation or a hydrograph (Q versus time) for an unsteady simulation. SRH-2D calculates a distribution of the velocity vector along the inlet in such a way that the total discharge is satisfied. Three approaches may be used for the velocity distribution at the inlet such that the total specified discharge is satisfied.

<u>Uniform-v Approach</u>: A constant velocity magnitude is imposed at the inlet with flow direction normal to the inlet boundary.

Uniform-q Approach: A constant unit discharge, $q=vh$, is assumed with flow direction normal to the inlet boundary (v is velocity magnitude and h is water depth at inlet).

Conveyance Approach: A conveyance parameter is calculated first such that $K = Q / \sum_i \frac{h_i^{5/3}}{n_i} \Delta s_i$ with i = the i-th boundary face of the inlet, h_i = the water depth, n_i = the Manning's coefficient, and Δs_i = i-th boundary face distance. The velocity at each face i is then calculated as $v_i = K h_i^{2/3} / n_i$. The flow direction is assumed to be normal to the inlet boundary.

Currently, flow direction not normal to the inlet boundary is not available.

If flow is subcritical at an inlet, water surface elevation at the inlet is not needed and is calculated by SRH-2D assuming that the water surface slope normal to the inlet is constant.

If flow is supercritical at an inlet, water surface elevation at the inlet is also needed as the boundary condition. Currently, only a constant water surface elevation may be specified.

If the k-ε turbulence model is used, k and ε values are needed at an inlet. For most applications, they are not important and have negligible impact on the flow pattern (Rodi, 1980). SRH-2D, therefore, uses default values based on the relationships proposed by Rastogi and Rodi (1978) at an inlet: $\upsilon_t = 0.0765 U_* h$ and $\varepsilon = g U_* S$, in which S is energy slope and U_* is the friction velocity. Or, the following k and ε values are specified at an inlet: $k = 0.922 U_*^2$ and $\varepsilon = U_*^3 / h$ with $U_*^2 = g n^2 (U^2 + V^2) / h^{1/3}$.

7.3 Exit Boundary

An exit boundary is defined as a boundary segment on the solution domain where flow is expected to move out of the domain. Multiple exits may be specified for a problem.

At an exit where the flow is expected to be subcritical, only the water surface elevation is needed as the boundary condition. No boundary conditions are needed if the flow at the exit is supercritical. SRH-2D will automatically calculate the variables at the exit assuming that derivatives of variables normal to the boundary are constant.

Several ways may be used to supply the water surface elevation condition at a subcritical exit and they are discussed below.

User-Specified Water Surface Elevation: A user may specify the water surface elevation (stage), steady or a time series, directly at an exit. The elevation may be constant or a function of time. Often, the water surface elevation at the exit is either from measured data or from a 1D model such as HEC-RAS or SRH-1D that includes a much larger spatial area of the simulation river reach.

Rating Curve Approach: A user may provide rating curve data that gives the water surface elevation at the exit as a function of the flow discharge. SRH-2D calculates the water surface elevation at the exit automatically based on the flow discharge through the exit.

Free Surface Elevation (to be completed): For unsteady simulation such as flood propagation, the free surface elevation condition may be used in which the water surface elevation at the exit is calculated by SRH-2D using the kinematics condition, i.e., $\frac{\partial h}{\partial t}+\sqrt{gh}\frac{\partial h}{\partial n}=0$ (*n* here refers to the unit normal at the boundary). Exit-averaged quantities are used to obtain the average water surface elevation across the exit.

7.4 Solid Wall Boundary

Solid wall boundaries may represent banks and islands. No-slip condition is assumed at solid walls for the dynamic wave solver. However, a solid wall is equivalent to symmetry boundary for the diffusive wave solver. Therefore, only no-slip wall condition for the dynamic waver solver is described below.

The wall function approach is employed at a solid wall. With this approach, the flow shear stress vector at a wall boundary face is calculated as follows:

$$(\tau_{wx},\tau_{wy})=\rho C_{\mu}^{1/4}k_P^{1/2}\frac{\kappa(U,V)}{\ln(Ey_P^+)} \tag{12}$$

with $y_P^+=C_{\mu}^{1/4}k_P^{1/2}y_P/\upsilon$ for $k-\varepsilon$ model; and

$$(\tau_{wx},\tau_{wy})=\rho U_*\frac{\kappa(U,V)}{\ln(Ey_P^+)} \tag{13}$$

with $y_P^+=U_*y_P/\upsilon$ for depth-averaged parabolic model (zero-equation model).

In the above, C_{μ} is defined in equation (10), k_P is turbulent kinetic energy at cell P that contains the wall boundary face, $\kappa=0.41$ is the von Karman constant, y_P is normal distance from cell center P to a wall, and E is a constant.

For the $k-\varepsilon$ model, P_h and ε at cell P are fixed and calculated as:

$$P_h = \tau_w^2 / (\kappa \mu y_P^+) \text{ and } \varepsilon = C_\mu^{3/4} k_P^{3/2} / (\kappa y_P) \tag{14}$$

At solid walls, the gradient of sediment concentration in the direction normal to a wall is set to zero.

7.5 Symmetry Boundary

Symmetry boundary is defined as a boundary where all dependent variables are extrapolated assuming the gradient of the variable in a direction normal to the boundary is zero except the velocity component normal to the boundary. The velocity component normal to the boundary is set to zero.

Note that the symmetry boundary acts the same as the slip wall boundary condition within SRH-2D.

CHAPTER 8

NUMERICAL METHODS

This chapter provides the numerical methods and algorithms used to solve the governing equations in Chapter 5. Refer to Lai (2010) for more details.

8.1 Flow Solver

8.1.1 Discretization

The 2D depth-averaged equations in (1) to (3) may be written in tensor form as

$$\frac{\partial h}{\partial t} + \nabla \bullet (h\vec{V}) = 0 \tag{15}$$

$$\frac{\partial (h\vec{V})}{\partial t} + \nabla \bullet (h\vec{V}\vec{V}) = -gh\nabla z + \nabla \bullet \left(h\vec{\vec{T}} \right) - \frac{\vec{\tau}_b}{\rho} \;\; \text{s} \tag{16}$$

where $\vec{V}$ is the mean velocity vector, $\vec{\vec{T}}$ is the 2nd-order tensor of turbulence stress with its component defined in equation (5), $\overrightarrow{\tau_b}$ is the bed shear stress vector, and ρ is the fluid density. Note that rainfall is omitted as it is used only for the diffusive wave equation.

The governing equations are discretized using the finite-volume approach, following the work of Lai (1997, 2000) and Lai et al. (2003a). The solution domain is covered with an unstructured mesh with each mesh element assuming arbitrarily shaped polygons. Most commonly used polygons are triangles and quadrilaterals. All dependent variables are stored at the geometric center of a polygon. The governing equations are integrated over a polygon using the Gauss theorem. As an illustration, consider the general convection-diffusion equation representative of all governing equations:

$$\frac{\partial h\Phi}{\partial t} + \nabla \bullet (h\vec{V}\Phi) = \nabla \bullet (\Gamma \nabla \Phi) + S_{\Phi}^{*} \tag{17}$$

Here Φ denotes any dependent variable, a scalar or a component of a vector, Γ is the diffusivity, and S_{Φ}^{*} is the source/sink term. Integration over an arbitrarily shaped polygon P shown in Figure 7 leads to:

$$\frac{(h_P^{n+1}\Phi_P^{n+1} - h_P^n\Phi_P^n)A}{\Delta t} + \sum_{all-sides}(h_C V_C |\vec{s}|)^{n+1}\Phi_C^{n+1} = \sum_{all-sides}(\Gamma_C^{n+1}\nabla\Phi^{n+1} \bullet \vec{n}|\vec{s}|) + S_\Phi \quad (18)$$

In the above, Δt is time step, A is polygon area, $V_C = \overrightarrow{V_C} \bullet \vec{n}$ is the velocity component normal to the polygonal side (e.g., P_1P_2 in Figure 7) and is evaluated at the side center C, $\vec{n}$ is polygon side unit normal vector, $\vec{s}$ is the polygon side distance vector(e.g., from P_1 to P_2 in Figure 7), and $S_\Phi = S_\Phi^* A$. Subscript C indicates a value evaluated at the center of a polygon side and superscript, n or n+1, denotes the time level. In the remaining discussion, superscript n+1 will be dropped for ease of notation. Note that the first-order Euler implicit time discretization is adopted. The main task of the discretization is to obtain appropriate expressions for the convective and diffusive fluxes at each polygon side.

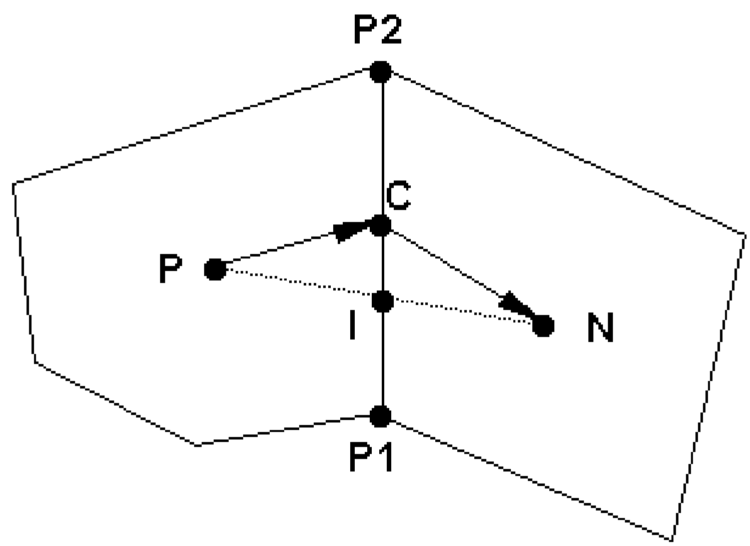


Figure 7. Schematic illustrating a polygon P along with one of its neighboring polygons N

Discretization of the diffusion term, the first term on the right hand side of equation (18), needs further attention. The final expression for $\nabla\Phi \bullet \vec{n}$ can be written as:

$$\nabla\Phi \bullet \vec{n}\,|\vec{s}| = D_n(\Phi_N - \Phi_P) + D_c(\Phi_{P2} - \Phi_{P1}) \quad (19)$$

where

$$D_n = \frac{|\vec{s}|}{(\vec{r}_1 + \vec{r}_2) \bullet \vec{n}} \; ; \qquad D_c = -\frac{(\vec{r}_1 + \vec{r}_2) \bullet \vec{s} \,/|\vec{s}|}{(\vec{r}_1 + \vec{r}_2) \bullet \vec{n}} \tag{20}$$

In the above, $\vec{r}_1$ is the distance vector from P to C and $\vec{r}_2$ is from C to N. The normal and cross diffusion coefficients, D_n and D_c, at each polygon side involve only geometric variables; they are calculated only once in the beginning of the computation.

Calculation of a variable, say *Y*, at the center C of a polygon side is discussed next. This is an interpolation operation used frequently for a number of variables. In the next, a second-order accurate expression is derived. As shown in Figure 7, a point I is defined as the intercept point between line PN and line P_1P_2. A second-order interpolation for point I gives:

$$Y_I = \frac{\delta_1 Y_N + \delta_2 Y_P}{\delta_1 + \delta_2} \tag{21}$$

in which $\delta_1 = \vec{r}_1 \bullet \vec{n}$ and $\delta_2 = \vec{r}_2 \bullet \vec{n}$. Y_I may be used to approximate the value at the side center C. This treatment, however, does not guarantee second-order accuracy unless $\vec{r}_1$ and $\vec{r}_2$ are parallel. A truly second-order expression is derived as:

$$Y_C = Y_I - C_{side}(Y_{P2} - Y_{P1}) \tag{22a}$$

$$C_{side} = \frac{(\delta_1 \vec{r}_2 - \delta_2 \vec{r}_1) \bullet \vec{s}}{(\delta_1 + \delta_2)|\vec{s}|^2} \tag{22b}$$

The extra term in the above is similar in form to the cross diffusion term in equation (20).

Φ_C in the convective term in equation (18) needs further discussion. If the second-order scheme is applied directly, spurious oscillations may occur for flows with a high cell Peclet number (Patankar 1980). Therefore, a damping term is added to the second-order scheme similar to the concept of artificial viscosity. The damped scheme is derived by blending the first-order upwind scheme with the second-order central difference scheme and can be expressed as

$$\Phi_C = \Phi_C^{CN} + d(\Phi_C^{UP} - \Phi_C^{CN}) \tag{23}$$

where

$$\Phi_C^{UP} = \frac{1}{2}(\Phi_P + \Phi_N) + \frac{1}{2} sign(V_C)(\Phi_P - \Phi_N) \tag{24}$$

and Φ_C^{CN} is the second-order interpolation scheme, equation (22a). In the above expression, *d* defines the amount of damping used. In most applications, *d* = 0.2 ~ 0.3 is used.

With expressions for the diffusion and convection terms, the final discretized governing equation for an element P can be organized as the following linear equation

$$\mathrm{A}_P \Phi_P = \sum_{nb} \mathrm{A}_{nb} \Phi_{nb} + S_{diff} + S_{conv} + S_\Phi \tag{25}$$

where "nb" refers to all neighbor polygons surrounding the polygon P. The coefficients in this equation are:

$$A_{nb} = \Gamma_C D_n + Max(0, -h_C V_C |\vec{s}|) \tag{26a}$$

$$A_P = \frac{h_P^n A}{\Delta t} + \sum_{nb} A_{nb} \tag{26b}$$

$$S_{diff} = \frac{h_P^n A}{\Delta t} + \sum_{all\ sides} \Gamma_C D_c \left(\Phi_{P2} - \Phi_{P1}\right) \tag{26c}$$

$$\begin{aligned} S_{conv} = &\sum_{all\ sides} (h_C V_C |\vec{s}|) \left\{ (1-d) \left[\frac{\delta_1}{\delta_1 + \delta_2} - \frac{1 - sign(V_C)}{2} \right] (\Phi_N - \Phi_P) \right\} \\ &- \sum_{all\ sides} (h_C V_C |\vec{s}|) \left[(1-d) C_{side} (\Phi_{P2} - \Phi_{P1}) \right] \end{aligned} \tag{26d}$$

8.1.2 Side Normal Velocity Calculation and Elevation Correction Equation

For a non-staggered mesh, a special procedure is required to obtain the polygon side normal velocity that is used to enforce the continuity equation. Otherwise the well-known checkerboard instability may appear (Rhie and Chow 1983). Here the procedure proposed by Rhie and Chow (1983) and Peric et al. (1988) is adopted. That is, the normal velocity is obtained by averaging the momentum equation from element centers to element sides. A detailed derivation is omitted, but interested readers are referred to the previous work (e.g., Rhie and Chow 1983, Peric et al. 1988, and Lai et al. 1995). It is sufficient to express the final side normal velocity as follows:

$$V_C = <\vec{V}> \bullet \vec{n} + \left\langle \frac{A}{A_P} \right\rangle < gh\nabla z > \bullet \vec{n} - \left\langle \frac{A}{A_P} \right\rangle gh\nabla z \bullet \vec{n} \tag{27}$$

where “< >” stands for the interpolation operation from mesh element center to side as expressed in (22a). When a vector appears in the interpolation operation, the interpolation is applied to each Cartesian component of the vector.

The velocity-water surface elevation coupling is achieved using a method similar to the SIMPLEC algorithm (Patankar 1980). In essence, if the elevation from a previous time step or iteration, z^n, is known, an intermediate velocity field, may be obtained by solving the linearized momentum equation:

$$A_P \vec{V}_P^* = \sum_{nb} A_{nb} \vec{V}_N^* - a\nabla z^n + \vec{S}_V \tag{28}$$

where a is a constant. Next, we seek corrections of velocity $\vec{V}' = \vec{V}^{n+1} - \vec{V}^*$ and elevation $z' = z^{n+1} - z^n$ such that the momentum equation is satisfied, i.e.,

$$A_P \vec{V}_P^{n+1} = \sum_{nb} A_{nb} \vec{V}_N^{n+1} - a\nabla z^{n+1} + \vec{S}_V \tag{29}$$

Or, the following correction equation is obtained:

$$A_P \vec{V}_P' = \sum_{nb} A_{nb} \vec{V}_N' - a\nabla z' \tag{30}$$

With the SIMPLEC algorithm, the above may be approximated as

$$\vec{V}_P' = -\frac{a}{A_P - \sum_{nb} A_{nb}} \nabla z' \tag{31}$$

Substitution of the above into the continuity equation (15) leads to the following elevation correction equation:

$$\frac{z'}{\Delta t} + \nabla \bullet (\vec{V} z') = \nabla \bullet \left(\frac{ah}{A_P - \sum_{nb} A_{nb}} \nabla z' \right) - \nabla \bullet \left(h^n \vec{V}^* \right) \tag{32}$$

The above elevation correction equation may be solved to obtain z' and then (31) is used to obtain the velocity correction. A number of iterations are usually needed within each time step if the flow is unsteady; but one iteration is used for a steady state simulation.

8.1.3 Summary of Solution Procedure

Governing equations are solved in an equation-by-equation manner. In a typical iterative solution process, momentum equations are solved first assuming known water surface elevation and turbulent viscosity given at the previous time step. The newly obtained velocity is used to calculate the normal velocity at mesh element sides in equation (27). This side velocity will usually not satisfy the continuity equation. Therefore, the pressure correction equation (32) is solved and (31) is used to obtain a new elevation and new velocity. After the elevation correction equation, other scalar equations, such as turbulence and sediment equations, may be solved. This completes one iteration of the solution cycle. The above iterative process may be repeated within one time step until a preset residual criterion for each equation is met. Then the solution would advance to the next time step. For a steady state simulation, one iteration is usually used as time-accurate intermediate solutions are usually not sought. In this study, the residual of a governing equation is defined as the sum of absolute residuals at all mesh elements.

The implicit solver requires the solution of non-symmetric sparse matrix linear equations in (25). Direct solvers are impractical for calculations with a lot of mesh elements because of excessive demand for computer memory and CPU time. ON the other hand, the choice of iterative solvers is limited for the unstructured mesh. In SRH-2D, the standard conjugate gradient solver with ILU preconditioning is used (Lai 2000).

CHAPTER 9

VERIFICATION CASES

This chapter focuses on verification of SRH-2D, as a numerical technique or model has to be tested to lend credence to its validity and application range. A number of verification and test cases are presented, from simple cases with analytical solutions to those with experiment data, and some are compared with published numerical results of other models. The next chapter presents application and validation cases when SRH-2D is applied to practical projects.

9.1 1D Subcritical Flow in a Channel

MacDonald (1996) presents a number of non-trivial analytical test cases for 1D steady St. Venant equations. Test case 1 is a subcritical flow that is selected to test the dynamic wave solver of SRH-2D. Case 1 has a horizontal extent of 1000m by 10m with a variable bed slope. A steady flow discharge of 15 m^3/s is maintained at the upstream boundary while a water depth of 0.7484m is maintained at the exit. The Manning's roughness coefficient used for simulation is 0.03 and the Froude number of the flow ranges from 0.40 to 0.77.

An 81-by-4 mesh is used to simulate the case as shown in Figure 8 with the boundary conditions of discharge at the inlet and water depth at the exit.

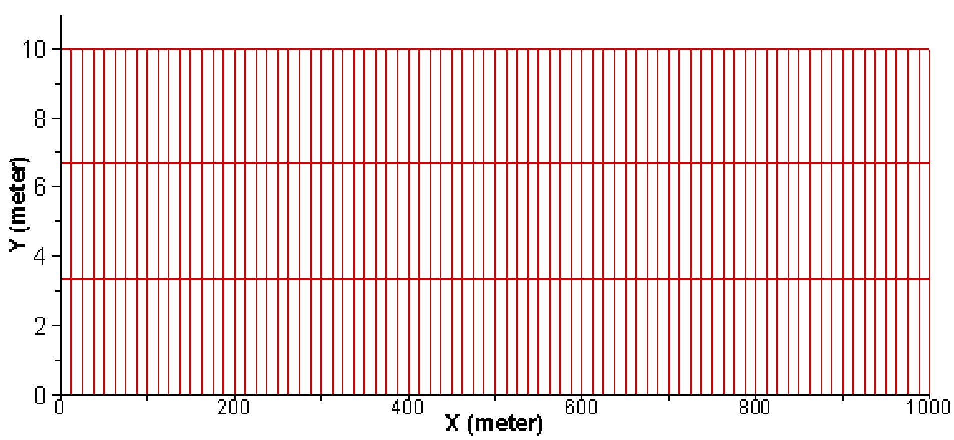


Figure 8. An 81-by-4 mesh used for simulation for test 1 of MacDonald (1996)

Simulated water surface elevation and water depth are compared with the analytical solution of MacDonald (1996) in Figure 9 and Figure 10. Also, the diffusive wave solver is used to simulate the case and results from the diffusive wave solver are also shown. It is seen that the simulated water surface elevation and water depth are almost the same as the analytical solution, while the water depth result of the diffusive wave solver produces slight errors.

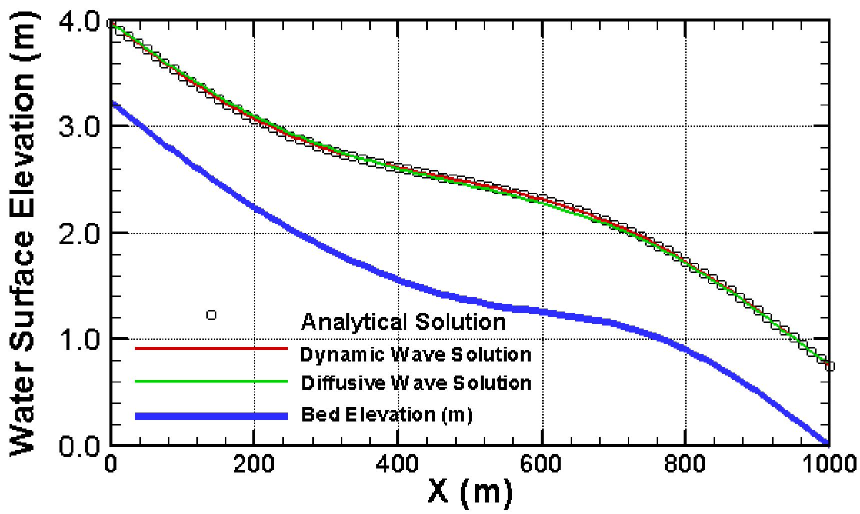


Figure 9. Comparison of simulated water surface elevation with analytical solution for test case 1 of MacDonald (1996)

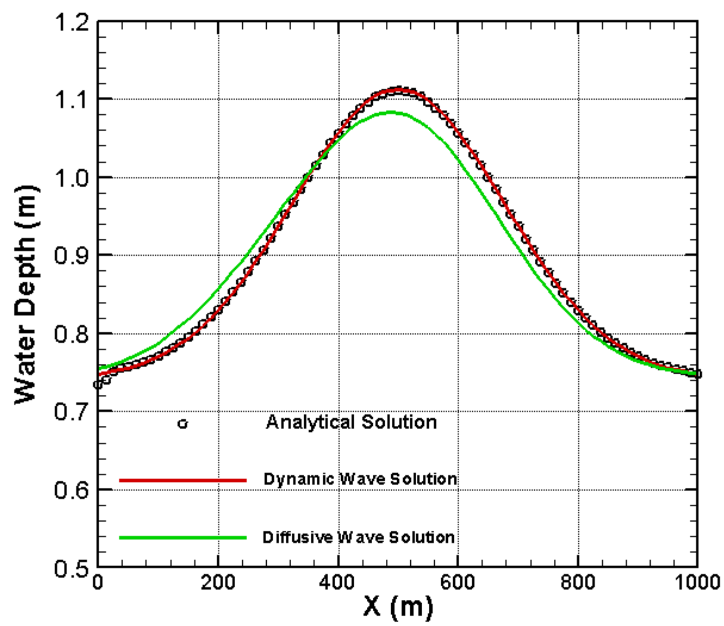


Figure 10. Comparison of simulated water depth with analytical solution for test case 1 of MacDonald (1996)

9.2 1D Transcritical Flow in a Channel

Test case 6 of MacDonald (1996) is a transcritical flow that has a smooth transition from subcritical to supercritical flow with a hydraulic jump. It is selected to test the dynamic wave solver of SRH-2D. The test case has a horizontal extent of 150m by 10m with a variable bed slope. A steady subcritical flow discharge of 20 m^3/s is maintained at the upstream boundary while a water depth of 1.7m is maintained at the exit. The Manning's roughness coefficient is 0.031752.

A 121-by-4 uniform Cartesian mesh is used that is similar to the 1D subcritical flow case in Figure 8. A discharge is specified at the subcritical inlet while water depth is specified at the subcritical exit.

Simulated results are plotted in Figure 11 that show the 3D view of the bed and water surface elevations with color of the water surface representing the Froude number. It is seen that the subcritical flow at the inlet quickly transitions to supercritical, and a hydraulic jump is then formed downstream. Simulated water surface elevation and water depth profiles are compared with the analytical solution of MacDonald (1996) in Figure 12 and Figure 13. Also, the diffusive wave solver is used to simulate the case and results from the diffusive wave solver are also shown. It is seen that the dynamic wave solution compares well with the analytical including capturing of the hydraulic jump. On the other hand, the diffusive wave missed the hydraulic jump completely and a smooth transition of the water surface elevation is simulated. This indicates that the diffusive wave solver is inappropriate for modeling hydraulic jumps. However, the simulated results of the diffusive wave solver is checked against the analytical solution of the diffusive wave equation. It is shown that a comparison between the model and the analytical results are quite good.

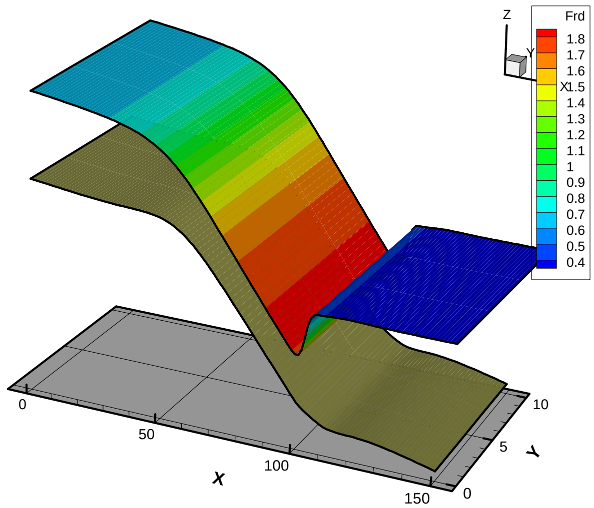


Figure 11. 3D view of bed elevation and simulated water surface elevation for test case 6 of MacDonald (1996)

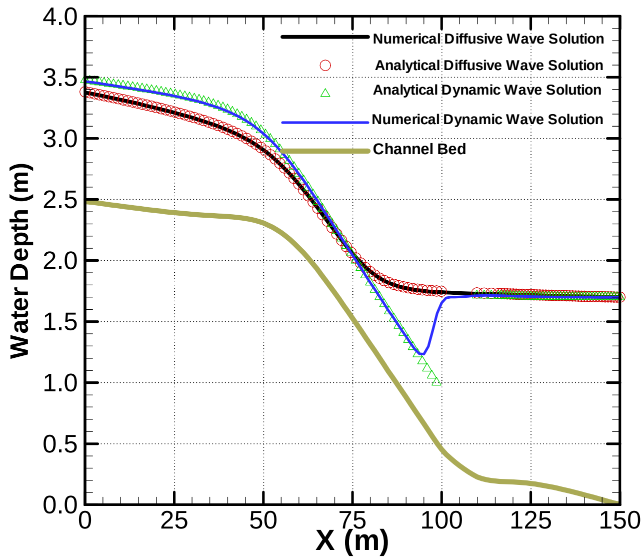


Figure 12. Comparison of simulated water surface elevation with analytical solution for test case 6 of MacDonald (1996)

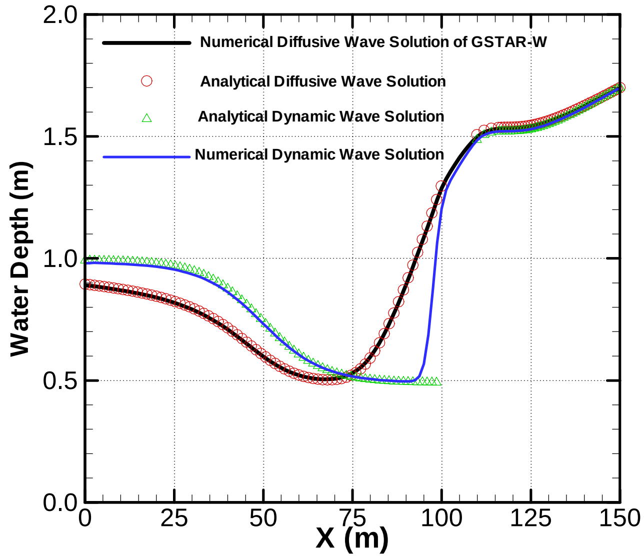


Figure 13. Comparison of simulated water depth with analytical solution for test case 6 of MacDonald (1996)

9.3 2D Diversion Flow in a Channel

A channel bifurcation occurs often in open channel flows, and flow features are complex in the diversion area. This test case simulates a channel diversion case measured and studied by Shetta and Murthy (1996). It serves as a 2D test case with flow separations. Results with k-e turbulence model are also presented by lai (2010) which gives much improved results.

The solution domain consists of a main channel, with 6.0m in length (*X* direction) and 0.3m in width (*Y* direction), and a side channel normal to the main channel at *X*=3.0m. The side channel has a length of 3.0m and width of 0.3m. A quadrilateral mesh system was used to cover the solution domain and the portion of the mesh at the diversion is shown in Figure 14, along with the *X* and *Y* coordinate system. Overall, the main channel has a mesh of 120-by -30 elements and the side channel has 40-by-30 mesh elements.

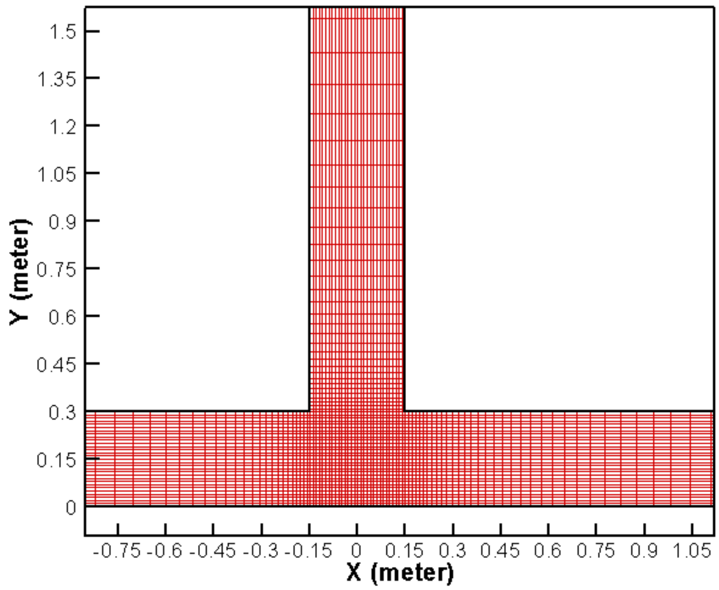


Figure 14. Part of the quadrilateral mesh used for simulation of the diversion flow

The simulated case has a main channel flow discharge of 0.00567 m^3/s, water surface elevation of 0.0555m at the exit of the main channel (X=6.0m), and water surface elevation of 0.0465m at the exit of the side channel (Y=3.3m). The Manning's roughness coefficient is 0.012 and the parabolic turbulence model is used for the simulation.

Simulated results are compared with measured data of Shettar and Murthy (1996) for the water surface elevation along both walls of the main and side channels (Figure 15 and Figure 16) and depth averaged velocity profiles in both channels (Figure 17 and Figure 18). The water surface elevation in the main channel is predicted well but discrepancy is noticeable in the side channel. Also, the velocity near the bottom wall (Y=0) of the main channel is over-predicted. These discrepancies, mostly associated with areas of flow separation, are due to the inability of the turbulence model to predict the size of flow separation accurately. Results may be improved with the use of the k-ε turbulence model instead of the parabolic model used.

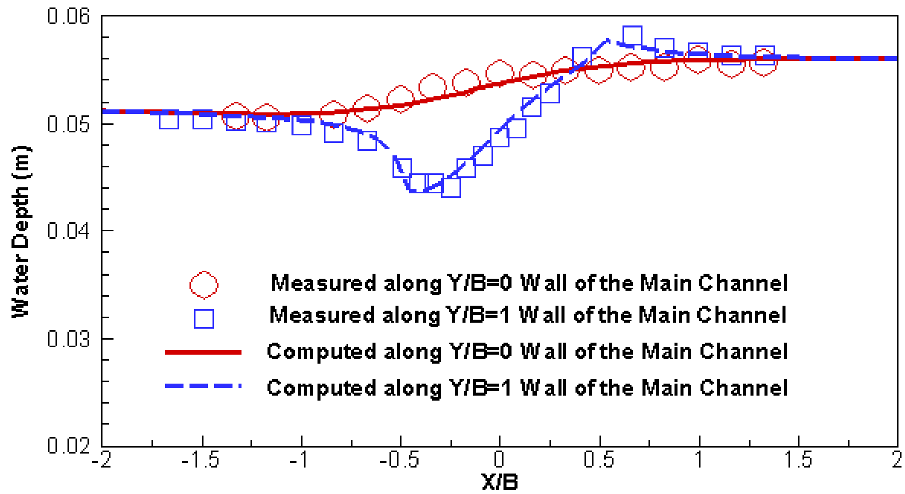


Figure 15. Comparison of water surface elevation along both walls of the main channel for the Shettar and Murthy (1996) case.

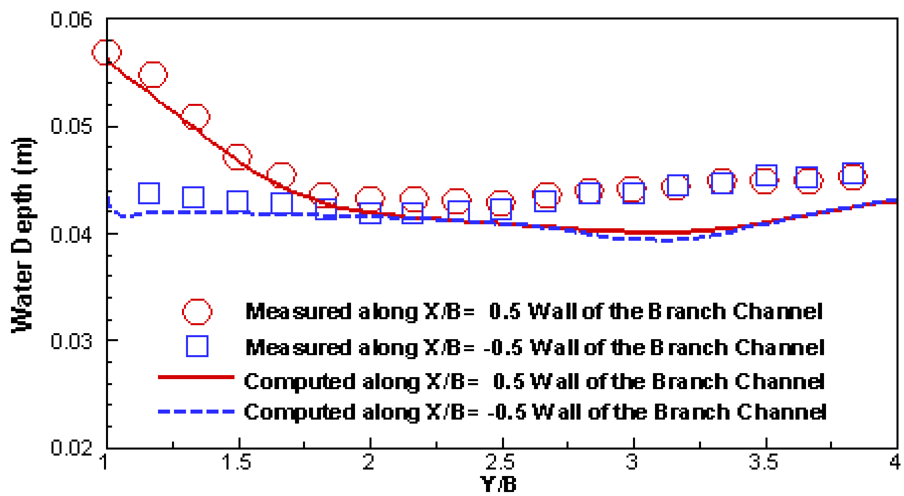


Figure 16. Comparison of water surface elevation along both walls of the side channel for the Shettar and Murthy (1996) case.

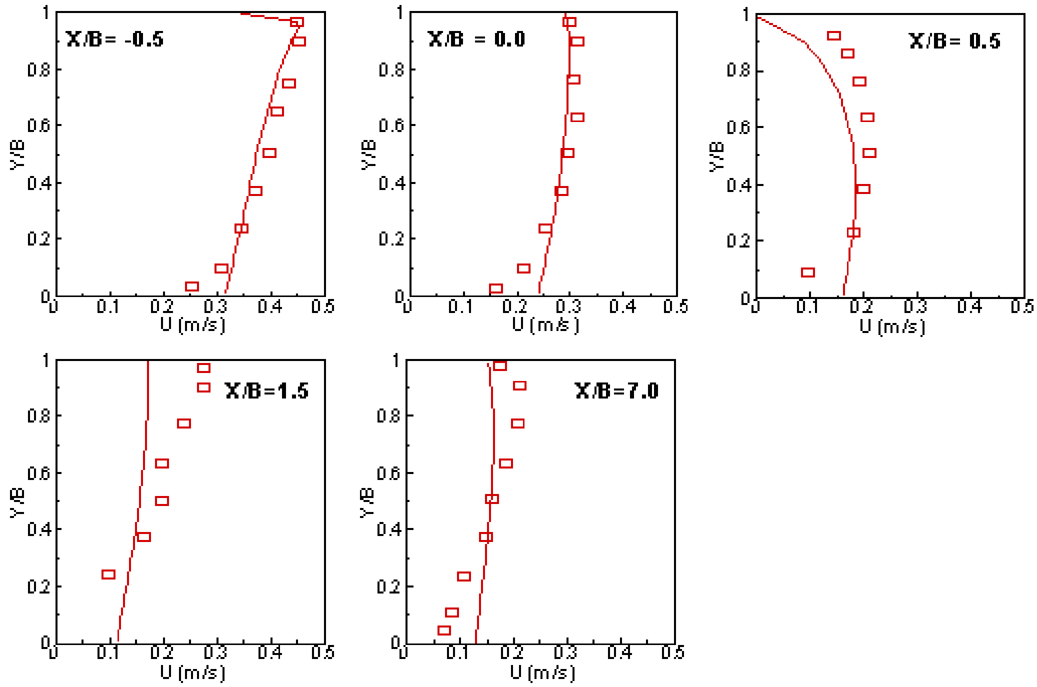


Figure 17. Comparison of x-velocity (U) profiles at selected x locations in the main channel for the Shettar and Murthy (1996) case.

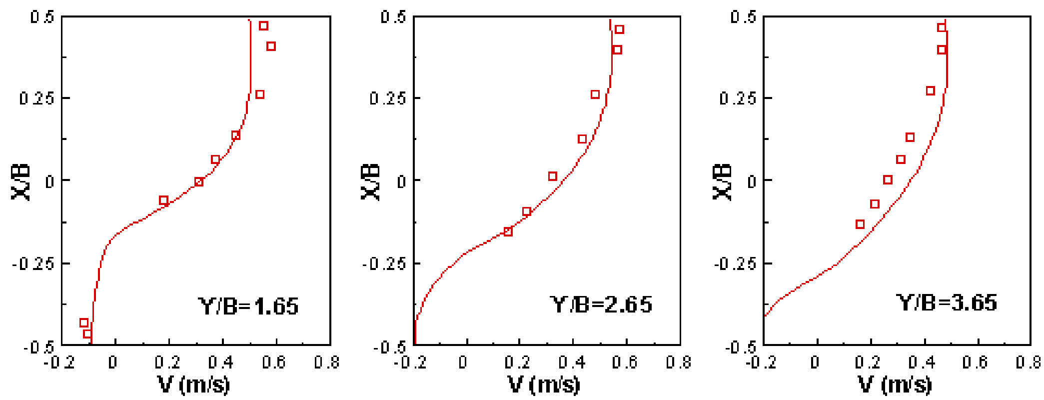


Figure 18. Comparison of y-velocity profiles at selected y locations in the side channel for the Shettar and Murthy (1996) case.

CHAPTER 10

APPLICATION CASES

SRH-2D has been applied to many projects for practical applications and this chapter focuses on presentation and discussion of selected applications and validation cases. Each case discussed has a separate project report that provides much more detail and the user is referred to the respective reports for further information.

10.1 Savage Rapids Dam Removal Study

This section presents application of SRH-2D to a dam removal study, the Savage Rapids Dam. The Savage Rapids Dam is located in southwestern Oregon on the Rogue River, five miles upstream from the city of Grants Pass. It is owned and operated by Grants Pass Irrigation District and has been used for diverting irrigation flows since 1921. The full removal of the dam and construction of a new pumping station are under design by the Bureau of Reclamation, due to lack of compliance of the existing fish ladders and screens to the current National Marine Fisheries Service criteria. SRH-2D is used to simulate various scenarios to provide design data and assistance. Only the calibration and verification study is reported below. Detailed application results of SRH-2D may be found in the project report by Bountry and Lai (2006). Additional discussion of results may be found in Bountry et al. (2006).

10.1.1 Topography and Mesh

The simulation reach extends from the Savage Rapids Park, 0.5 mile upstream of the dam, to about 0.45 mile downstream of the dam. The topography for the reach is reconstructed from a number of surveys conducted between 1999 and 2005 (Bountry and Randle 2003) (see Figure 19). A quadrilateral mesh is developed that consists of 20,145 elements and 20,468 nodes with a typical element size of 5 by 12 feet. A 3D view of the topography and part of the mesh is displayed in Figure 20.

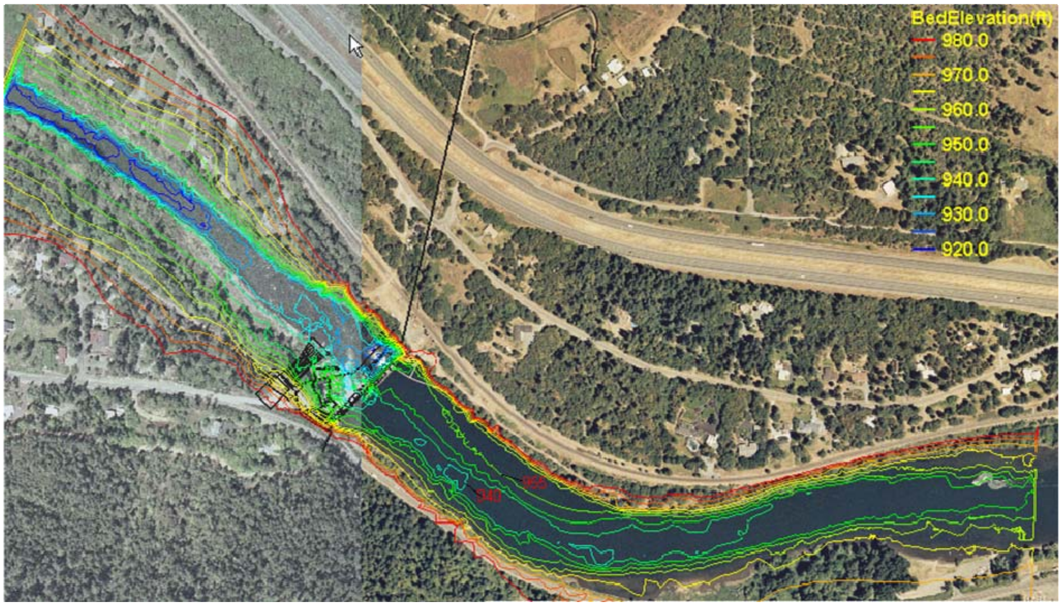


Figure 19. Plainview and bed elevation contours of the simulated area for the Savage Rapids Dam removal project

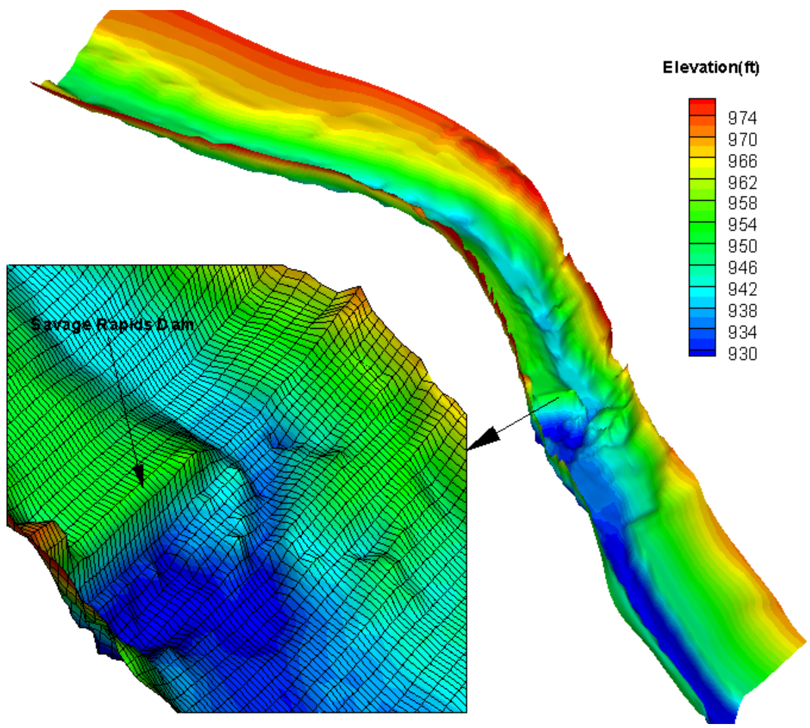


Figure 20. A Perspective View of the Topography of the Modeled River Reach.

10.1.2 Case Modeled, Boundary Conditions, and Other Parameters

The measured data, water surface elevation and velocity vectors, during the April 2002 survey (Bountry and Randle 2003) was chosen to calibrate and verify the SRH-2D model. This case represents a drawn-down flow with a discharge of 2,800 ft^3/s. All flow was through the two radial gates near the left side of the dam. The measured water surface elevation is used to calibrate the Manning roughness coefficient that is assumed to be uniform throughout the reach. Once calibrated, the model results are then compared with the measured velocities and flow patterns. Both diffusive wave and dynamic wave solutions are obtained so that a comparison may be made between the two solvers.

A water surface elevation of 935.53ft was specified at the downstream boundary. This elevation was obtained from the calibrated one dimensional HEC-RAS model as described by Bountry and Randle (2003). At the upstream boundary, a flow discharge of 2,800ft^3/s was applied where a uniform distribution of velocity is assumed with the flow normal to the boundary. The calibrated flow loss coefficient is 0.05 for the diffusive wave model and 0.04 for the dynamic wave model. Finally, the depth-averaged parabolic model is used for the turbulence viscosity used by the dynamic wave model (Rodi 1993).

10.1.3 Comparison of Water Surface Elevation

The calibrated model results are compared with the measured water surface elevation along the thalweg in Figure 21. Both the diffusive wave and the dynamic wave model agree with the measured elevation well. Major discrepancy between the two models is mostly limited to an area near the radial gates where a hydraulic jump exists due to the dam. As anticipated, the dynamic wave model predicts the existence of the jump, while the diffusive wave model is incapable of simulating the hydraulic jump. The diffusive wave model tends to predict a smooth variation of elevation over the jump. Based on experiences with other applications of SRH-2D, it is recommended that the jump area should be modeled with a higher loss coefficient in order to predict the water elevation change, although the uniform coefficient works fine for the Savage Rapids Dam application.

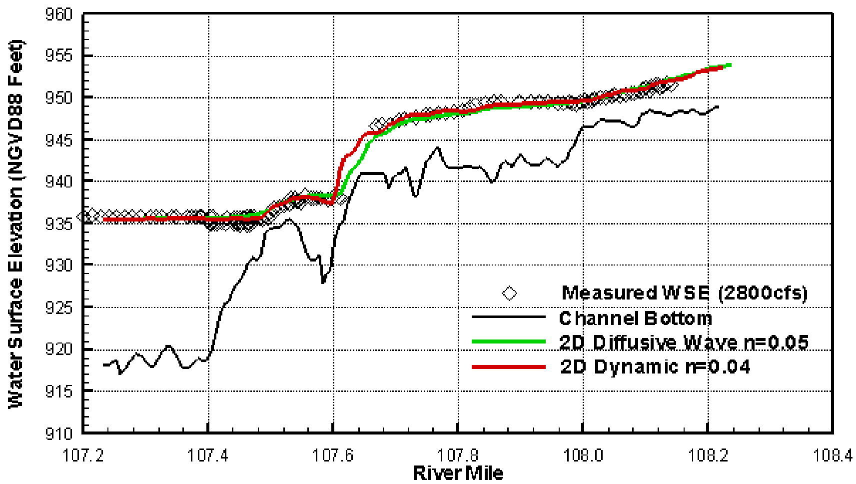


Figure 21. Comparison of Predicted and Measured Water Surface Elevations

10.1.4 Comparison of Velocities and Flow Patterns

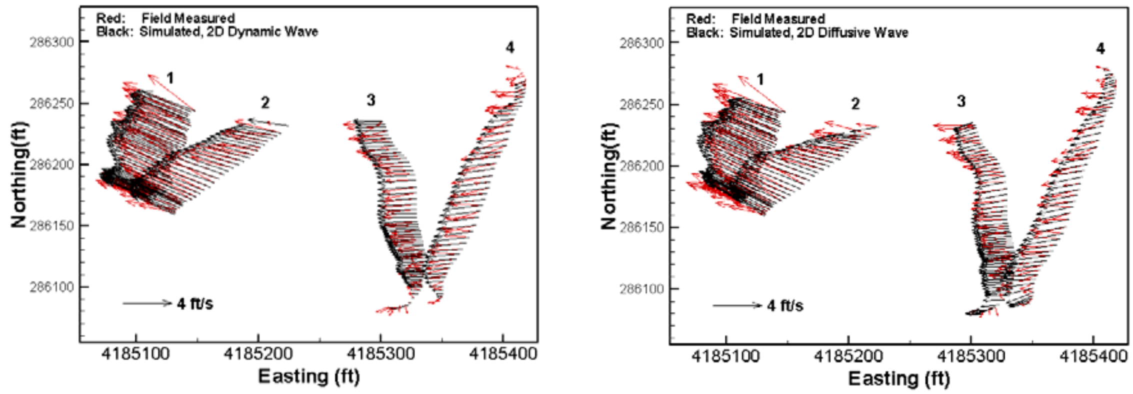
Next, the computed velocity vectors and flow patterns are compared with the measured data so that the flow hydraulics may be compared in greater detail. It is noted that a good prediction of the water surface elevation does not guarantee a good prediction of velocities and flow patterns.

The ADCP-measured depth-averaged velocity data are available and the measurement points are displayed in Figure 22. Upstream of the dam, eight cross sections were surveyed and they are numbered consecutively in the figure. Downstream of the dam, two areas are compared: One is immediately downstream of the dam but near the right side; another is downstream of the excavated channel from the radial gates. Complex eddies were formed at the time of the survey in both areas.

Figure 22. Velocity Measurement Points for the Simulated River Reach (Points are Shown in Red)

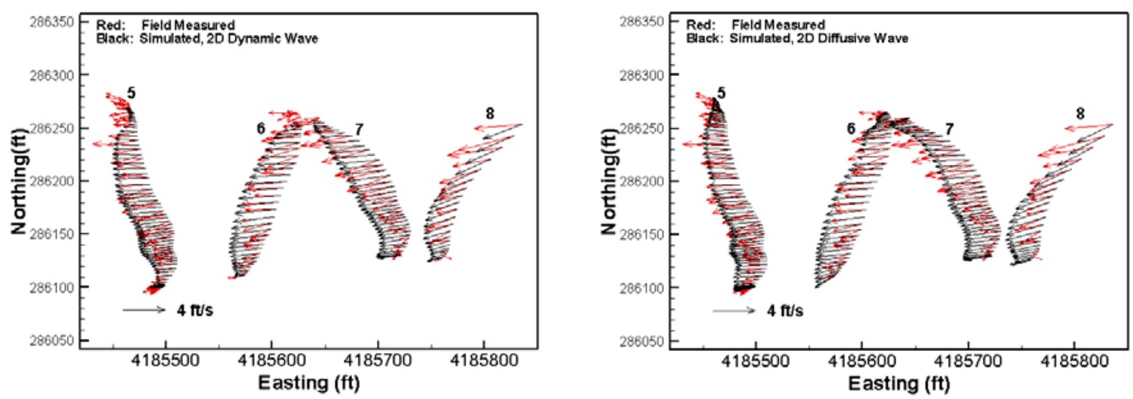
A comparison of predicted and measured velocity vectors at eight cross sections upstream of the dam is displayed in Figure 23 and Figure 24. Agreement is favorable for both models except at a few locations. Overall, the difference between the dynamic wave and the diffusive wave solutions is not appreciable. The dynamic wave model is capable of predicting the flow separation on the left bank of cross sections 3 and 4 while the diffusive wave model is not.

A comparison of velocities and flow patterns is shown downstream of the dam in Figure 25. It is clear that the diffusive model is incapable of predicting any eddies and therefore, the velocity results in such areas are in gross error. On the other hand, the dynamic wave model is quite good in predicting the eddy structures. It is noted that the two-eddy structures on the right of the jet stream from the excavated channel is well predicted both in terms of size and location. In addition, the eddy on the left of the jet stream is also predicted. These results indicate that the dynamic wave model has to be used if eddies or flow separation are of interest.



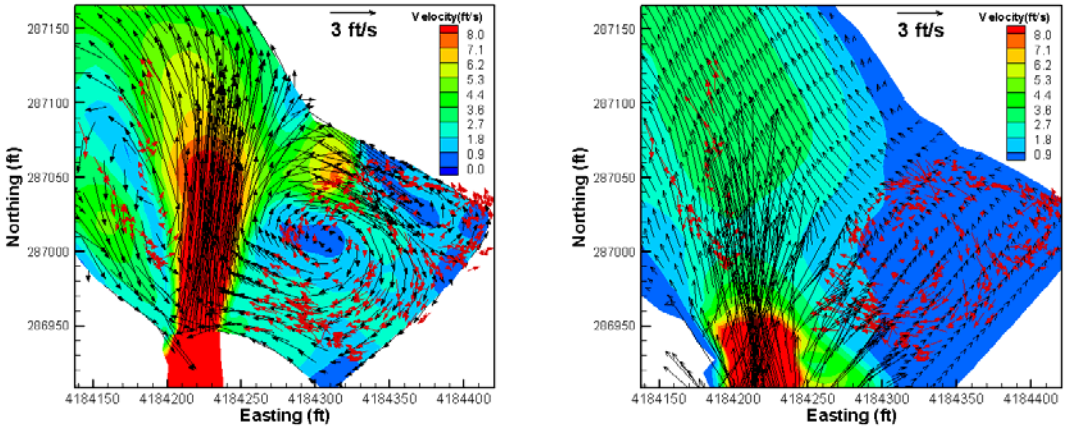
(a) Dynamic Wave Solution (b) Diffusive Wave Solution

Figure 23. Comparison of Predicted and Measured Velocity Vectors at Cross Sections 1 to 4



(a) Dynamic Wave Solution (b) Diffusive Wave Solution

Figure 24. Comparison of Predicted and Measured Velocity Vectors at Cross Sections 5 to 8



(a) Dynamic Wave Solution (b) Diffusive Wave Solution

Figure 25. Comparison of Velocity Vectors and Flow Patterns downstream of the Dam

10.2 Study of Sandy River and Columbia River Interaction

The Sandy River Delta Dam (SRD Dam) is located near the confluence of the Sandy and Columbia Rivers, east of Portland, Oregon. As a result of its closure in 1938 to improve fish passage through the Sandy River, flow has been redirected from the east (upstream) distributary to the west (downstream) distributary of the delta. The east distributary has since partially filled with sediment and supports dense riparian vegetation, including aged cottonwoods. Although once the main distributary channel, the east distributary is currently only activated under high flow conditions on the Sandy or Columbia Rivers. The study area is shown in Figure 26.

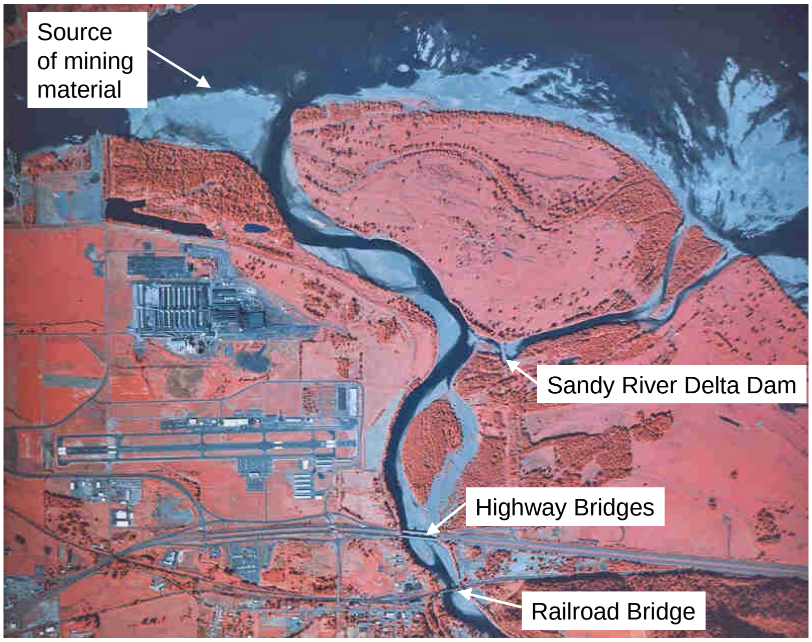


Figure 26. Aerial photo of the study area at the Sandy River Delta

Increased understanding of the ecological functions of the natural channel configuration and requirements of anadromous fish has initiated a reassessment of the role of the SRD Dam in improving fish passage. Recent efforts to improve aquatic habitat conditions have considered the removal of the SRD Dam. SRH-2D was used to more effectively evaluate possible effects related to removal of the SRD Dam. Both hydraulic and sediment studies were carried out but only the hydraulic results of the model calibration study are discussed. More details of the study may be found in the project report by Lai et al. (2006).

10.2.1 Solution Domain, Mesh, and Flow Roughness

The solution domain was selected based on the stated objectives of the project and was guided later by the topographic and bathymetric data; it is displayed in Figure 27. The solution domain encompassed about 9.5 miles of the Columbia River and 2.6 miles of the Sandy River with an area of about 12.8 square miles.

The final mesh is displayed in a series of figures from Figure 28 to Figure 30. A combination of quadrilateral and triangular elements was used that provided the best compromise between the accuracy and computing time. The main river channels were mostly covered with quadrilateral cells that allow mesh stretching while the remaining areas were mostly covered with combined triangular-quadrilateral cells. The final mesh contained a total of 37,637 cells.

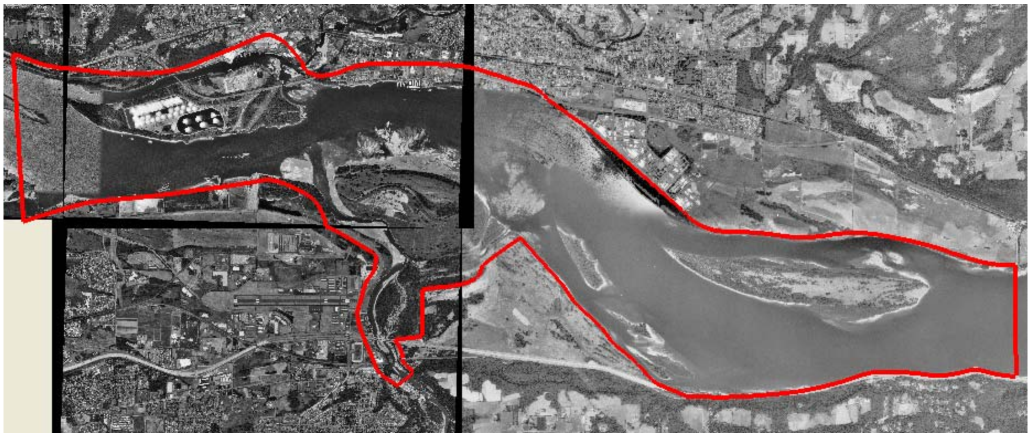


Figure 27. Solution domain for the Sandy River Delta simulation. West (left) side of the Columbia River is the exit boundary, east (right) side is the inlet boundary, and south (bottom) side is the inlet boundary of the Sandy River

Figure 28. Mesh for the Sandy River Delta project: entire solution domain.

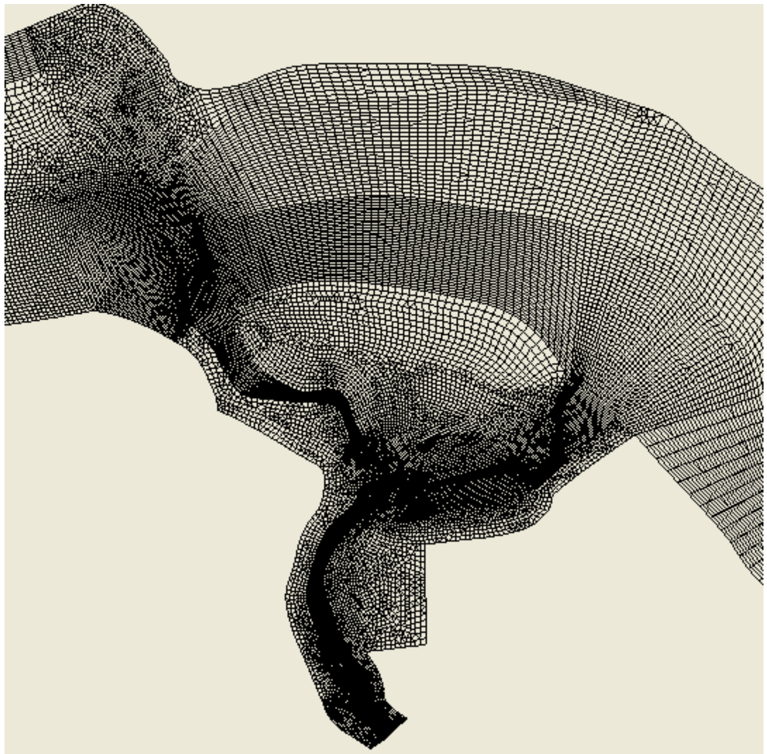


Figure 29. Mesh for the Sandy River Delta project: the Sandy River Delta area

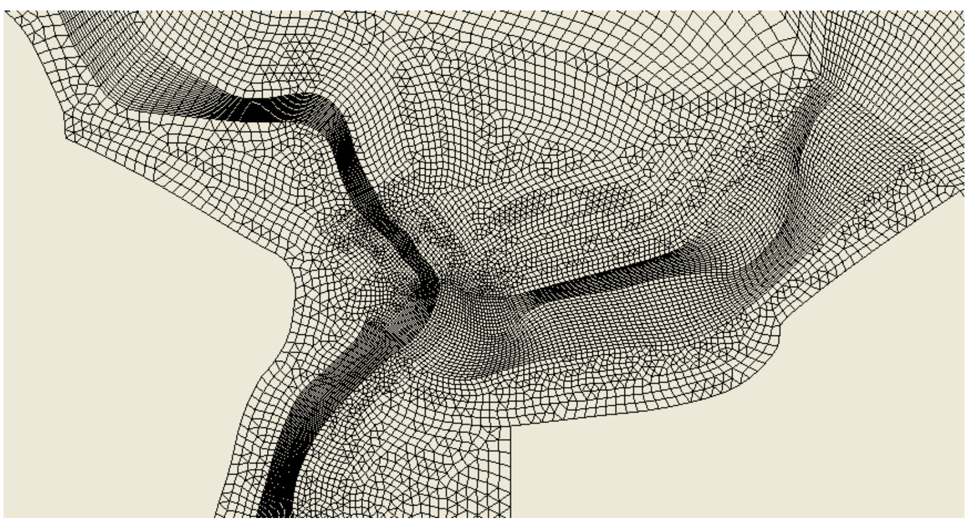


Figure 30. Mesh for the Sandy River Delta project: Dam area.

Topography data were obtained from several sources, including Lidar data and cross section survey data, to represent existing conditions. The bathymetric data were in point form (Easting, Northing, and elevation) and were interpolated onto the mesh points. The bed elevation contour plot and a perspective view of the topography are shown in Figure 31 and Figure 32.

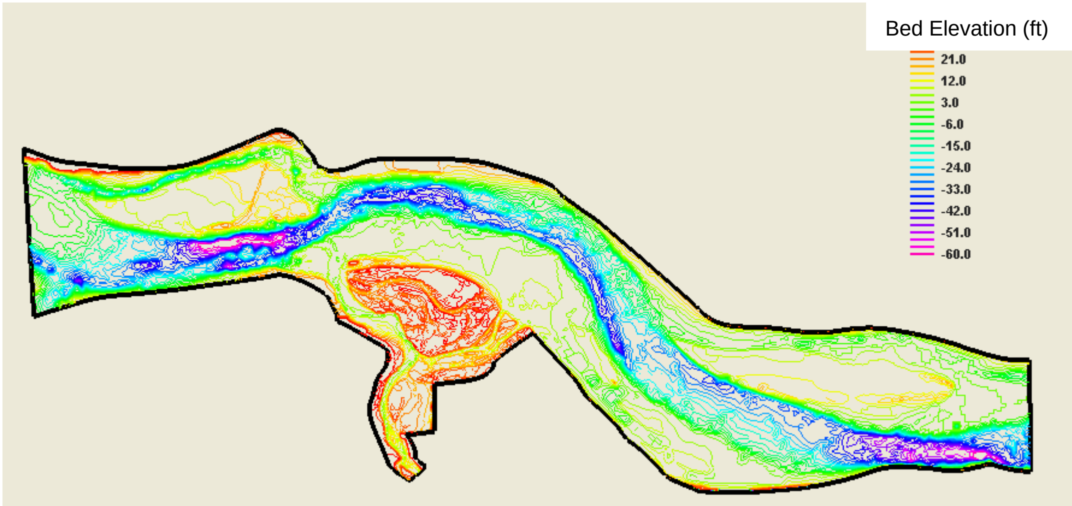


Figure 31. Contour plot of the bed elevation for the Sandy River Delta project

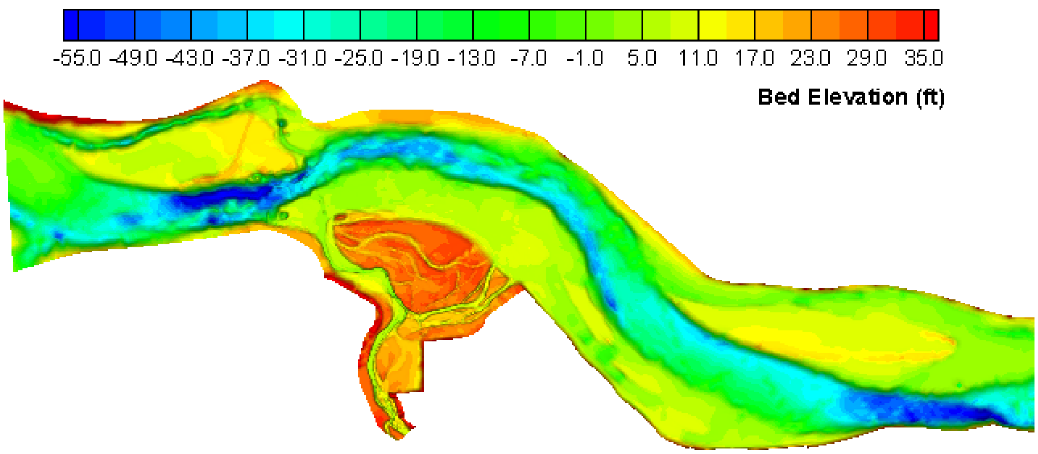


Figure 32. 3D perspective view of the topography for the solution domain.

Flow resistance was calculated with the Manning's roughness equation in which the Manning's coefficient (n) was needed as the model input. In this project, the solution domain was divided into a number of roughness zones as shown in Figure 33 according to the underlying bed properties. Note that zones 1, 2 and 3 represent the main channel of the Sandy River, and zones 4 and 5 represent the main channel of the Columbia River. Zone 6 consists mostly of sand bars and less vegetated areas, while zone 7 represents islands and floodplains with more vegetation. Each zone was assigned a Manning's n value that was determined through a calibration study by comparing with the field data of October 2005. After a number of simulation runs, the final calibrated Manning's coefficients were listed in Table 1.

Table 1. Calibrated Manning's Coefficients in Different Zones Shown in Figure 33

Zone Number	1	2	3	4	5	6	7
Manning's *n*	0.035	0.06	0.15	0.035	0.035	0.035	0.06

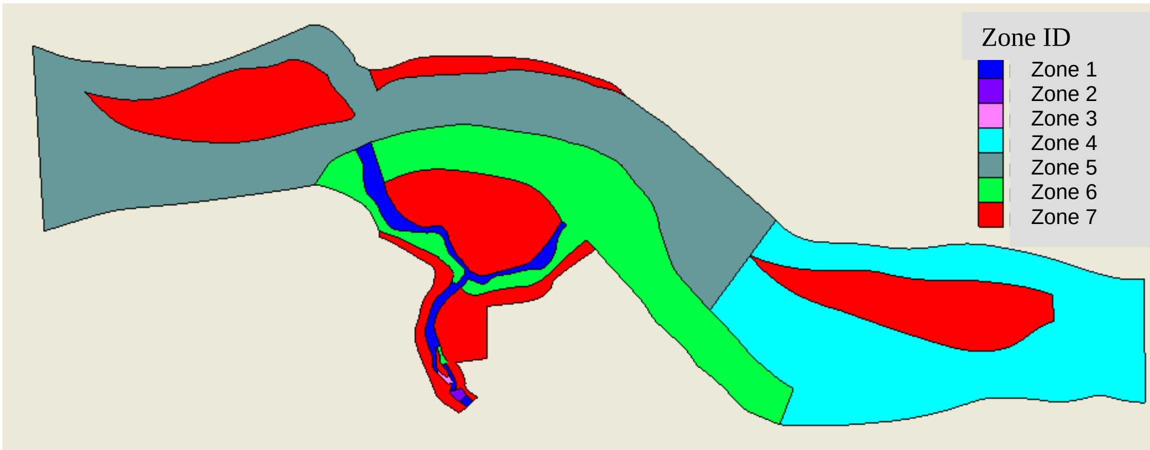


Figure 33. Roughness zones used for the Sandy River Delta Project

10.2.2 Input Data

Existing condition simulation was carried out corresponding to the field measured condition on October 12, 2005. The trip of October 2005 indicated that flow conditions were quite unsteady for both the Columbia River and the Sandy River, due mainly to the tidal influence and flow release from the Bonneville Dam. Flow unsteadiness often leads to difficulty in model calibration. Following a careful examination of the field data, conditions corresponding to the trip of October 12, 2005, were used for calibration.

The following input data were used for the model calibration study:

- Flow discharge for the Sandy River was set at 377 cfs, as recorded at the USGS Gage #14142500 (Sandy River below Bull Run River, near Bull Run, OR) on October 12, 2005. At one cross section of the Sandy River, field data from October 2005 estimated that the discharge was about 342 cfs based on the ADCP bottom tracking data.

- Flow discharge through the Columbia River was fixed at 123,000 cfs, which represented the average flow release from the Bonneville Dam on October 12, 2005. Releases from Bonneville Dam that day were very unsteady with a reported range of 118,000 to 132,000 cfs. Discharges calculated at several Columbia River cross sections from measured ADCP bottom tracking velocity data ranged from 98,310 to 125,700 cfs.

- The water surface elevation at the exit of the Columbia River reach was needed as the downstream boundary condition and the field measured elevation was used. The measured elevation, however, was quite unsteady and two distinct elevations were identified: 4.75 feet and 5.50 feet. Both elevations were used for the model calibration, which led to the development of two calibration runs: one low elevation case (4.75 feet) named Run #1, and another high elevation case (5.50 feet) named Run #2. Post-simulation analysis indicated that the difference in elevation at the exit boundary only influenced results near the confluence area of the Sandy River and Columbia River.

10.2.3 Comparison of Water Surface Elevation

Three sets of results were obtained with the calibrated hydraulic model, and they are named Run #1, #2 and #3. Run #1 and Run #2 reflect effects due to different water surface elevations specified at the exit boundary of the Columbia River reach. The two runs also indicate the sensitivity of model results to the exit boundary condition. Run #1 used the low elevation condition (4.75 feet), and Run #2 was based on the high elevation condition (5.50 feet). Both Run #1 and Run #2 used a Manning's coefficient of 0.15 for zone 3 in Figure 33. A third run (Run #3) was added to examine the impact of using a different Manning's coefficient in zone 3. Run #3 used the same downstream boundary condition as Run #1, but used a Manning's coefficient of 0.08 in zone 3 (versus 0.15 with Run #1 and #2).

The simulated water surface elevations on the Sandy River project reach are compared with the field data of October 2005 in Figure 34. The following observations may be made:

- The hydraulic model predicted the water surface elevation along the Sandy River quite well despite uncertainty in measured data and the unsteady nature of the flow in the field. The thalweg profile was also plotted in Figure 34 to demonstrate how well the model predicted water surface elevation despite large fluctuations in the bed topography. The difference between the field-measured and model-predicted elevation was typically within 0.3 feet, except near the confluence of the west distributary of the Sandy River and the Columbia River. This difference at the west confluence is likely associated with tidal fluctuations during the survey of October 2005.
- Major elevation changes at riffle and pool areas of the Sandy River reach were also predicted by the model. This indicates that the bed topography represented the riffle and pool areas correctly and that the model also represented the flow loss correctly.
- Uncertainty in the value of the Manning's *n* at Zone 3 may be obtained with results of Run #3. Reducing *n* from 0.15 to 0.08 alone led to a drop in

water surface elevation upstream of the zone by about 0.65 feet for the calibrated case. It should be noted that model-predicted elevations in other parts of the reach are not affected by this change. This assures that uncertainty in the roughness of zone 3 is limited to zone 3 only. A Manning's roughness coefficient of 0.08 was used when the model was applied to flood flow scenarios.

Comparison of water surface elevations on the Columbia River reach are shown in Figure 35. Again on the Columbia River, the river flow was quite unsteady and two distinct water surface elevations were identified. When different water surface elevations were used as the exit boundary conditions, represented by Run #1 and Run #2, the SRH-2D model predicted water surface elevations within the range of the measured values. Comparison of the field-measured and model-predicted water surface elevations demonstrates a satisfactory agreement along the Columbia River reach.

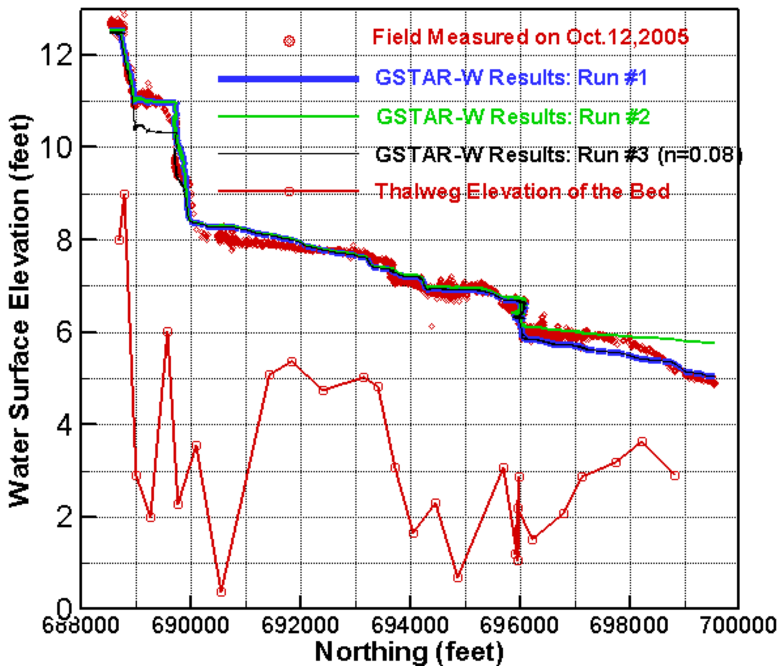


Figure 34. Comparison of simulated and field-measured water surface elevations along the Sandy River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D)

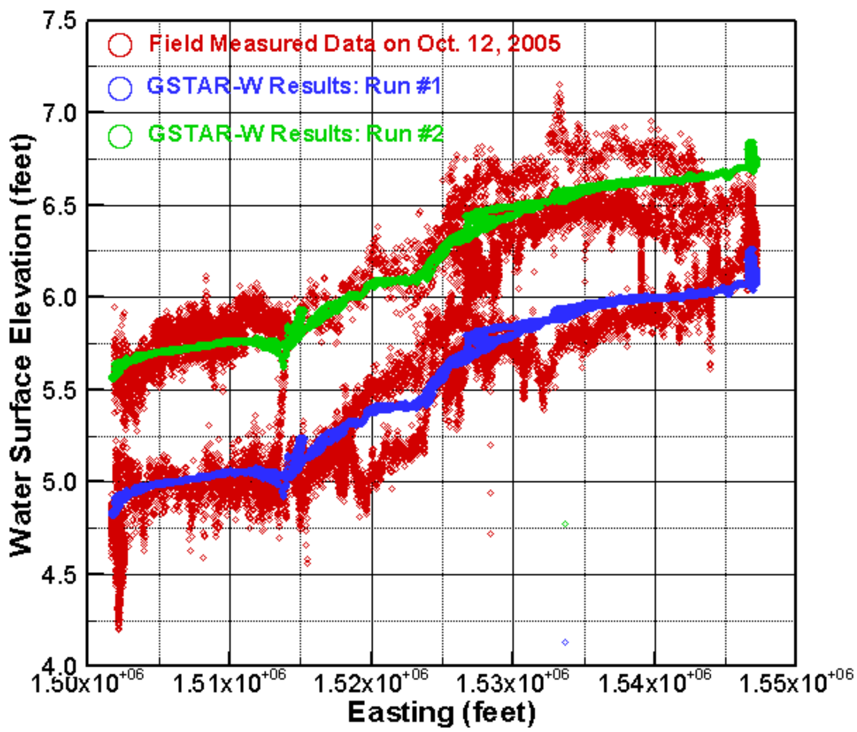


Figure 35. Comparison of simulated and field-measured water surface elevations along the Columbia River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D)

10.2.4 Comparison of Flow Velocity

Verification of the model was further carried out by comparing predicted and field-measured velocity results. ADCP measured velocity data were collected along both the Sandy and Columbia Rivers. An ensemble of ADCP data is a combination of water velocity (profile) and bottom tracking (boat velocity) data, and can be comprised of an average of several water velocity pings and several bottom pings. A ping is a single pulse of acoustic energy. Sandy River depth-averaged velocity data were processed from the ADCP velocity profiles (Water Mode 12) with 12 sub-pings. The Columbia River depth-averaged velocity data were from a single ADCP ensemble (velocity profile).

In both rivers, a measured data point represents an instantaneous, depth-averaged velocity for a single location. As a result, the data can be noisy, and averaging several adjacent velocity profiles is recommended in some situations. Research indicates that spatial averaging, sampling time, and sampling frequency affects the accuracy of mean velocity estimates (González-Castro *et al.*, 2000). However, no averaging of the field data was performed in this study for comparison with the model results, as we were only interested in evaluating if the simulated data fell within the range of measured data. An effort was made to remove all extreme outlier velocity data from the field-measured dataset.

Nevertheless, the dataset may still contain some erroneous data points (as can be seen from several velocity vectors presented). This does not affect the model calibration, but may contribute to a portion of the observed noise in the field-measured data.

Field-measured and model-predicted velocity magnitude comparisons at all measurement points were made for both the Sandy River (Figure 36) and the Columbia River (Figure 37). Although field data were noisy, results of the comparison are quite satisfactory. The large fluctuations in measured velocity values may be attributed to flow unsteadiness created by local geometry features, such as boulders and large turbulent eddies, and partly due to a few erroneous field data points.

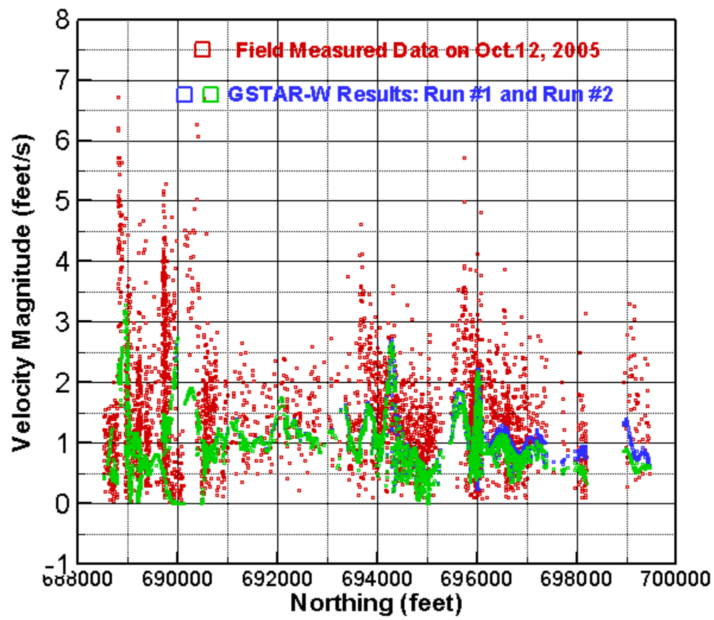


Figure 36. Comparison of simulated and field-measured velocity magnitudes along the Sandy River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D)

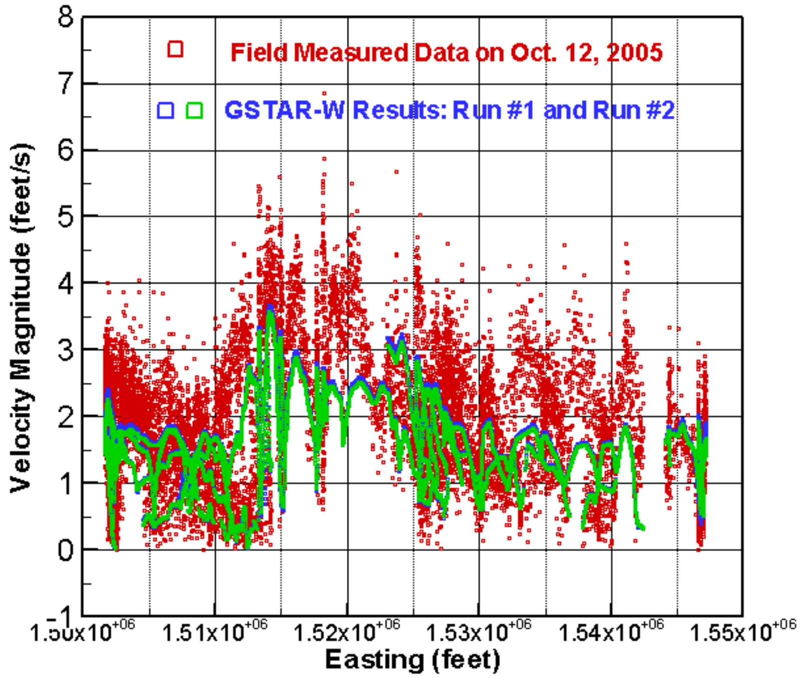


Figure 37. Comparison of simulated and field-measured velocity magnitudes along the Columbia River reach for October 12, 2005 flow conditions (GSTAR-W is the former name of SRH-2D)

Comparison of velocity was achieved through assessment of velocity vectors in different regions of the river reaches. Seven regions were used for comparison (Figure 38) and results are shown in Figure 39 to Figure 45. In view of uncertainty associated with some of the field data, the comparison between the field-measured and model-predicted data is deemed satisfactory.

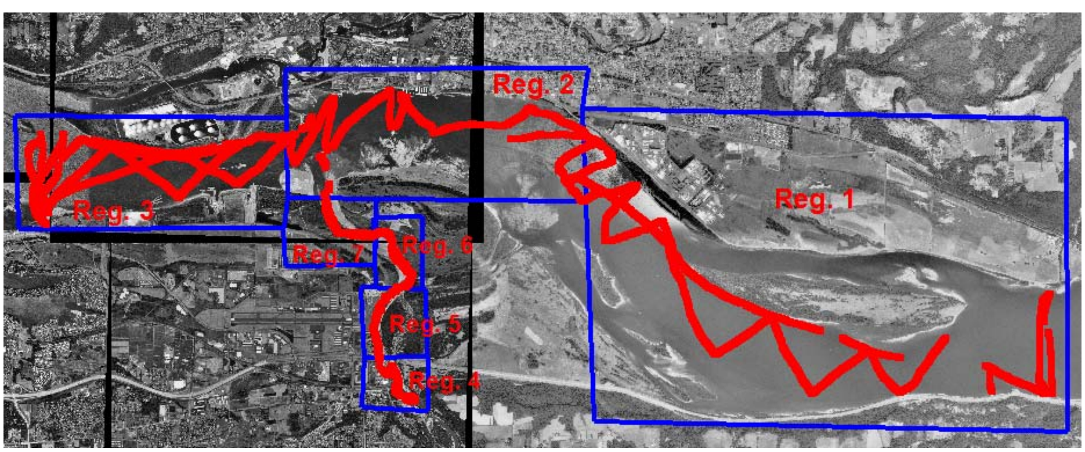


Figure 38. Seven regions (blue boxes) used for velocity vector comparison; Red points are the locations where velocity measurements were made

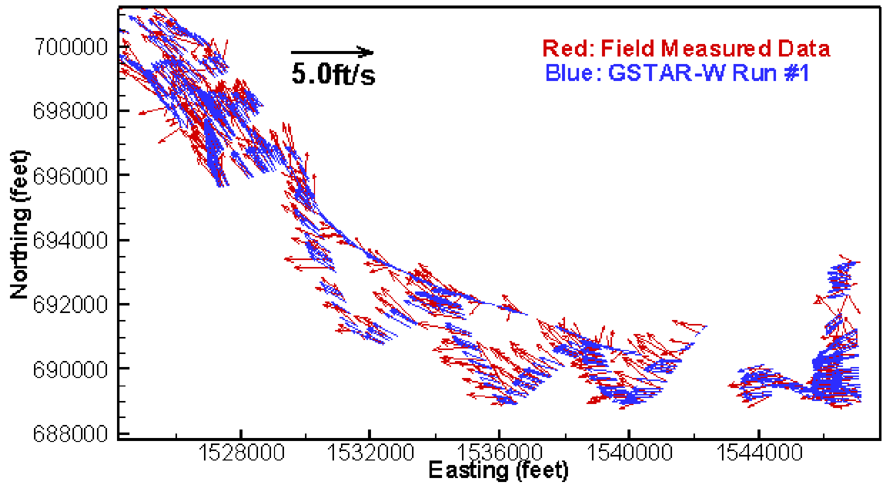


Figure 39. Comparison of velocity vectors in Region 1 (GSTAR-W is the former name of SRH-2D)

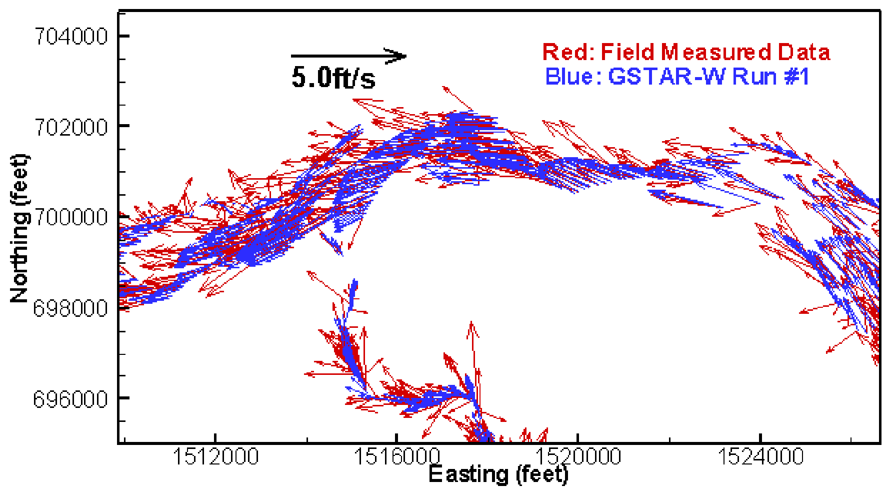


Figure 40. Comparison of velocity vectors in Region 2 (GSTAR-W is the former name of SRH-2D)

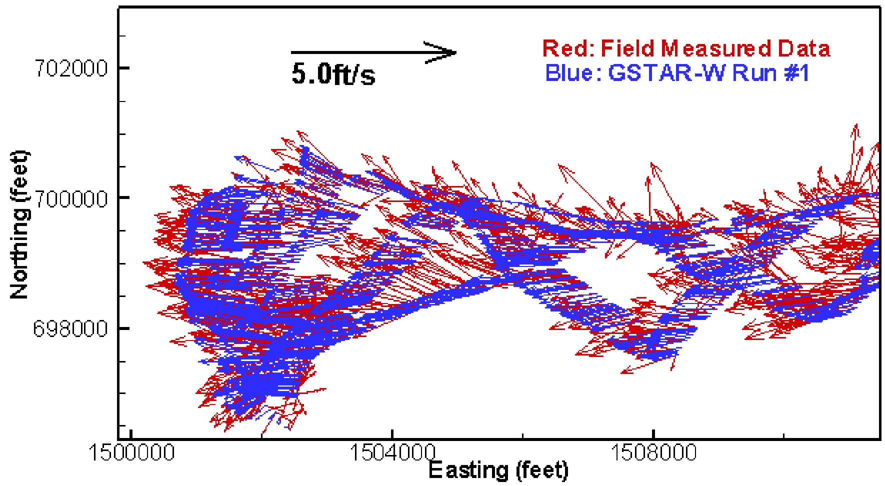


Figure 41. Comparison of velocity vectors in Region 3 (GSTAR-W is the former name of SRH-2D)

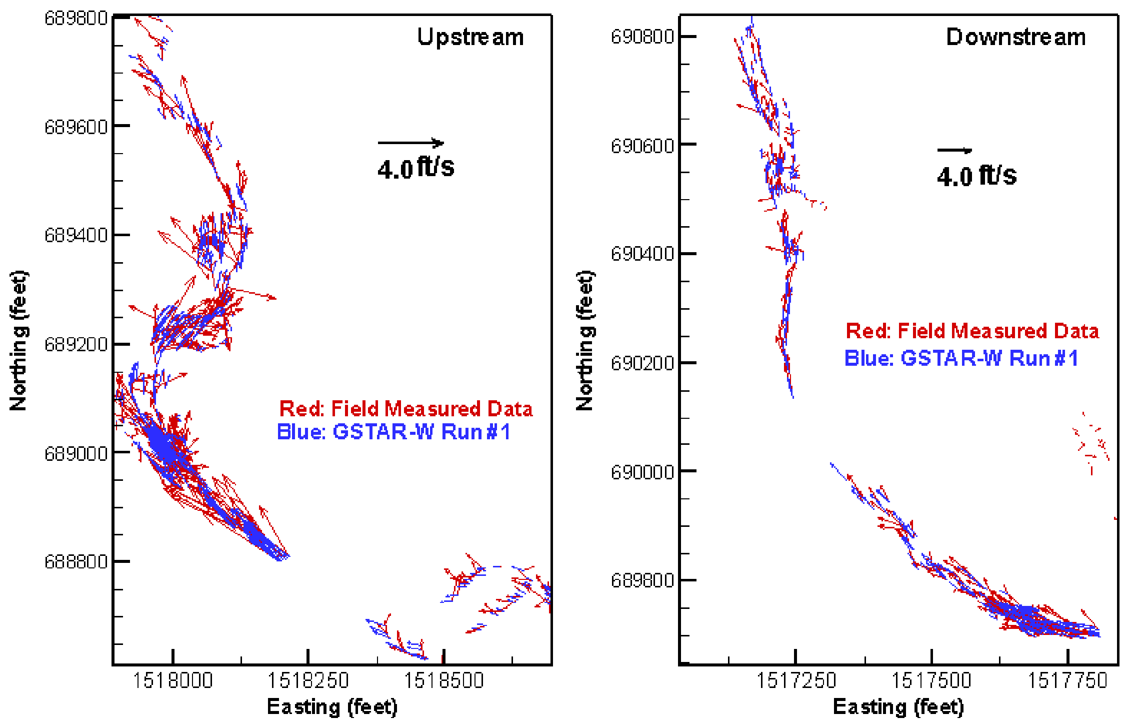


Figure 42. Comparison of velocity vectors in Region 4: Left is upstream and right is downstream portion of the region (GSTAR-W is the former name of SRH-2D)

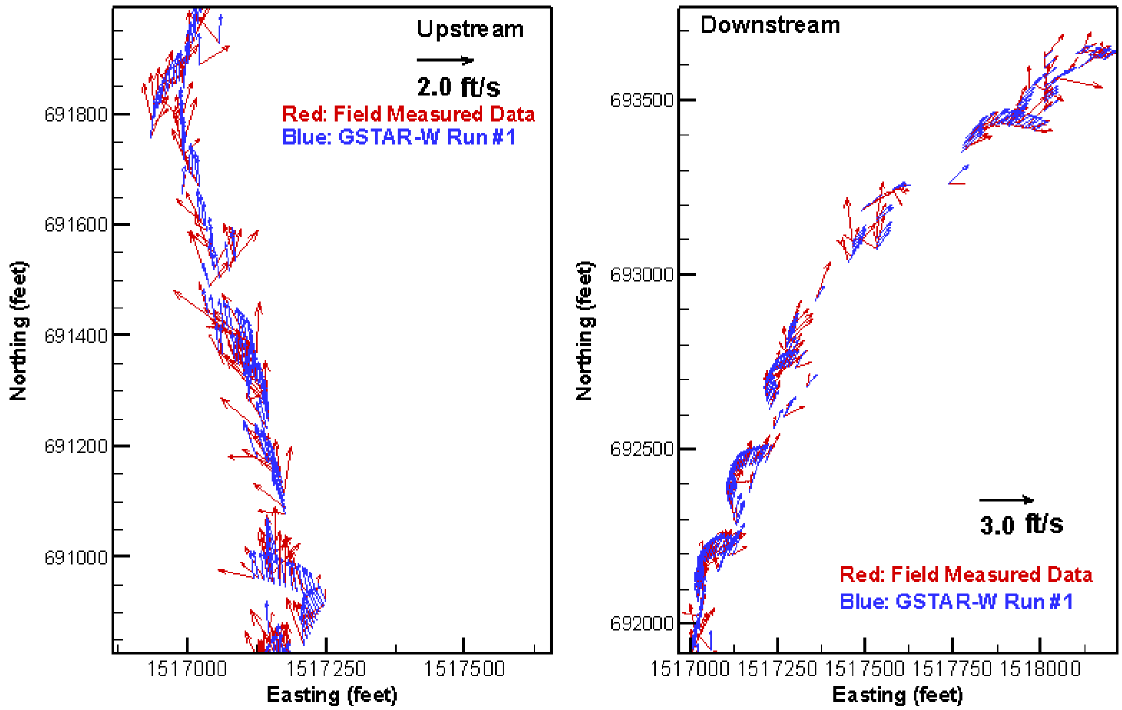


Figure 43. Comparison of velocity vectors in Region 5: Left is upstream and right is downstream portion of the region (GSTAR-W is the former name of SRH-2D)

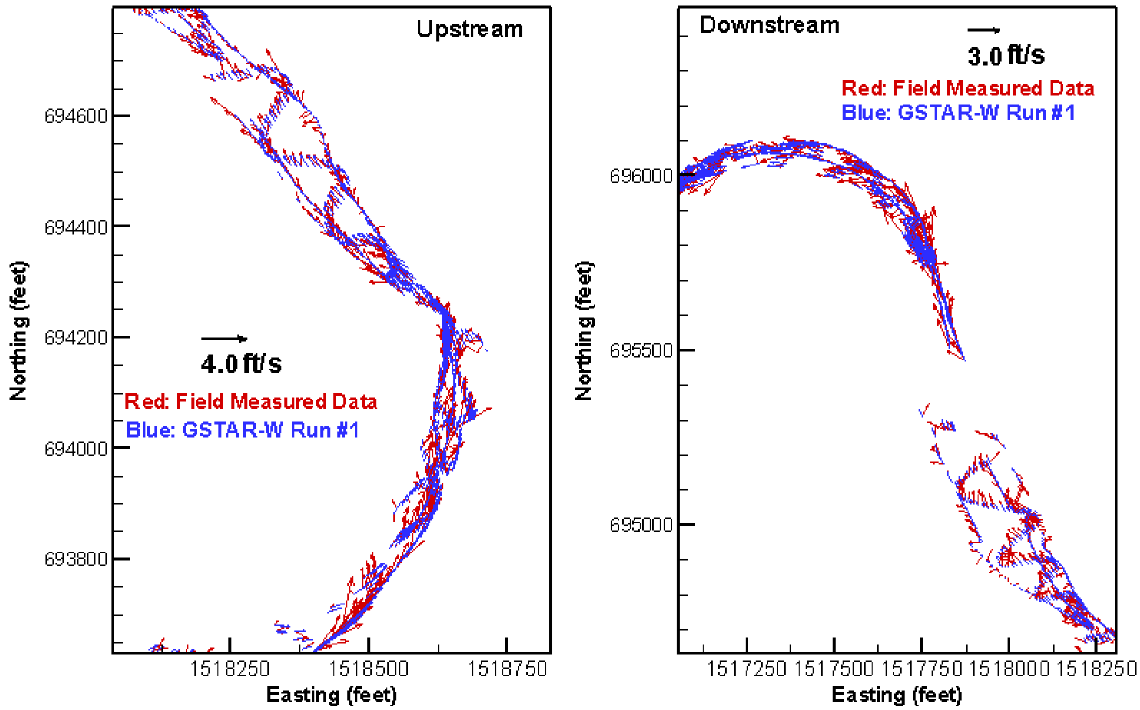


Figure 44. Comparison of velocity vectors in Region 6: Left is upstream and right is downstream portion of the region. (GSTAR-W is the former name of SRH-2D)

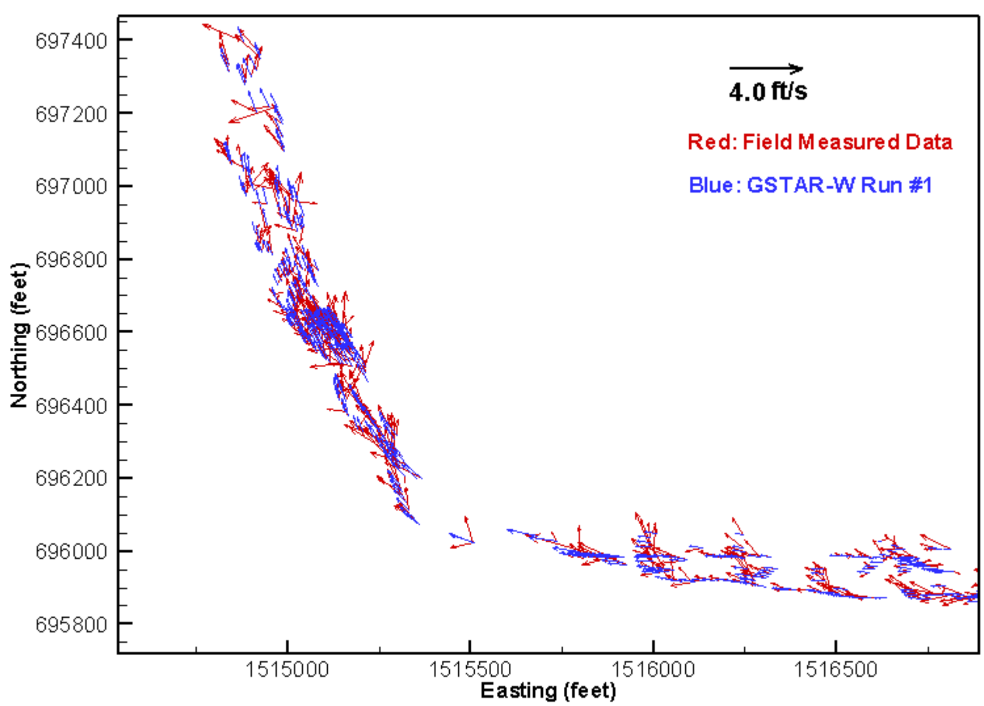


Figure 45. Comparison of velocity vectors in Region 7 (GSTAR-W is the former name of SRH-2D)

10.3 Other Application Cases

In addition to the projects discussed in this manual, SRH-2D has also been applied to many other projects with a wide range of applications. These include temporary diversion channel design, levee setback, stream habitat, dam removal, erosion and geomorphic assessment, etc. The following is a list of available project reports carried out at Reclamation up to 2008 for additional applications using the SRH-2D:

2008:

- **Columbia River Basin (Washington)**: Report: "Big Valley Reach Assessment, Methow River, Washington," Cassie Klumpp, 2008.
- **San Joaquin River (California)**: Report: "Two-Dimensional Modeling of the San Joaquin River: Reach 2B," Elaina Holburn, Blair Greimann, and Robert Hilldale, 2008.
- **Ventura River (California)**: Report: "Two-Dimensional Numerical Model Study of Sediment Movement at the Robles Diversion Dam on the Ventura River, California," Y. Lai and B. Greimann, 2008.
- **Nason Creek Geomorphic Assessment (Washington)**: Final Report: "Nason Creek Tributary Assessment," J. Bountry et al., 2008.
- **Lower Dungeness Restoration Project (Washington).** Final Report: "2D Hydraulic Modeling to Assist Dungeness Estuary Bluff Stability Assessment," Y. Lai and J. Bountry, 2008.
- **Middle Fork John Day River (Oregon):** "2D hydraulic modeling of the Forrest Conservation Property," E. Holburn, 2008.
- **Middle Fork John Day River (Oregon):** "2D hydraulic modeling of the Oxbow Conservation Property," Toni Turner, 2008.

2007:

- **Lower Dungeness Restoration Project (Washington).** Final Report: "Numerical Modeling Study of Levee Setback Alternatives for Lower Dungeness River, Washington," Y. Lai and J. Bountry, 2007.
- **Middle Rio Grande River (New Mexico).** Report: "Erosion Analysis Upstream of the San Acacia Diversion Dam on the Rio Grande River," Y. Lai, 2007.
- **Sacramento River (California):** Final Report: "Calibration of the Models for the Physical River Processes and Riparian Habitat on Sacramento River, California," B. Greimann et al., 2007.
- **Lower Colorado River (Arizona/California):** Final Report: "Bank Erosion Assessment Upstream of the Palo Verde Diversion Dam on the Lower Colorado River," Y. Lai, 2007.
- **Yakima River Basin (Washington):** Report: "Proposed Rehabilitation for the Schaake Reach of the Yakima River, Washington," R. Hilldale, 2007.

2006:

- **Savage Rapids Dam Removal Project (Grants Pass, Oregon).** Final Report: "Numerical Modeling of Flow Hydraulics in Support of the Savage Rapids Dam Removal," J. Bountry and Y. Lai, 2006.
- **Elwha Surface Diversion Project (Washington).** Final Report: "Numerical Hydraulic Modeling and Assessment in Support of Elwha Surface Diversion Project," Y. Lai and J. Bountry, 2006.
- **Sandy River Dam Removal Project (Troutdale, Oregon).** Final Report: "Analysis of Sediment Transport Following Removal of the Sandy River Delta Dam," Y. Lai and E. Holburn, 2006.
- **Yakima River Basin (Washington):** Report: "Identifying Stream Habitat with a Two-Dimensional Model – Report to the Yakima River Basin Water Storage Feasibility Study, Washington," R. Hilldale and D. Mooney, 2006

Recent papers, related to SRH-2D research and development, are listed below:

- Lai, Y.G., "Two-Dimensional Depth-Averaged Flow Modeling with an Unstructured Hybrid Mesh," J. Hydraulic Engineering, 136(1), 12-23, 2010.

- Lai, Y.G., Greimann, B., and Wu, K., "Predicting Rock Scour in an Alluvial River with a Two-Dimensional Model," ASCE World Environmental and Water Resources Congress, Kansas City, MI, May 17-21, 2009.

- Lai, Y.G. and Mooney, D., "On a Two-Dimensional Temperature Model: Development and Verification," ASCE World Environmental and Water Resources Congress, Kansas City, MI, May 17-21, 2009.

- Lai, Y.G., "Watershed Runoff and Erosion Modeling with a Hybrid Mesh Model," J. Hydrological Engineering, vol.14(1), 2009.

- Greimann, B., Lai, Y.G., and Huang, J., "A Total Load Equation for Sediment Transport Modeling," J. Hydraulic Engineering, vol.134(8), pp.1142-1146, 2008.

- Lai, Y.G. and Greimann, B., "Predicting Contraction Scour with a 2D Model" ASCE World Environmental and Water Resources Congress, Honolulu, HI, May 12-16, 2008.

- Lai, Y.G. and Greimann, B., "Modeling of Erosion and Deposition at Meandering Channels" ASCE World Environmental and Water Resources Congress, Honolulu, HI, May 12-16, 2008.

- Lai, Y.G. and Greimann, B., "Numerical Modeling of Alternate Bar Formation Downstream of a Dike." ASCE World Environmental and Water Resources Congress, Tampa, FL, May 15-19, 2007.

REFERENCES

Bountry, J.A. and Randle, T.J. (2003). "2D numerical model results at proposed intake sites for full dam removal," Project Final Report, Technical Service Center, Bureau of Reclamation, Denver, CO.

Bountry, J.A. and Lai, Y.G. (2006). "Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." Project Final Report, Technical Service Center, Bureau of Reclamation, Denver, CO.

Bountry, J.A., Lai, Y.G., and Randle, T.J. (2006). "Comparison of numerical hydraulic models applied to the removal of Savage Rapids Dam near Grants Pass, Oregon", 8th Federal Interagency Conference, Reno, NV, April 2-6, 2006.

Chow, V. T., Maidment, D. R. and Mays L. W. (1988). *Applied Hydrology*. New York: McGraw-Hill.

DHI Software (1996). "MIKE 21 Hydrodynamic Module Users Guide and Reference Manual," Danish Hydraulic Institute - USA, Eight Neshaminy Interplex, Suite 219, Trevose, PA. (Check the following website: http://www.dhisoftware.com/MIKE21/)

Eagleson, P. S. (1970). *Dynamic hydrology*, McGraw-Hill.

Flokstra, C. (1976). "Generation of two-dimensional horizontal secondary currents." *Rep. No. S163, Part II*, Delft Hydraulics Laboratory, Delft, The Netherlands.

González-Castro, J.A., Oberg, K., and J.J. Duncker (2000). "Effect of temporal resolution on the accuracy of ADCP measurements," *Proceedings of the ASCE 2000 Joint Conference on Water Resource Engineering and Water Resources Planning and Management, July 30- August 2, 2000*, Minneapolis, Minnesota, edited by R.H. Hotchkiss and M. Glade, 9 pp.

Lai, Y.G., So, R.M.C., and Przekwas, A.J. (1995). "Turbulent transonic flow simulation using a pressure-based method," *Int. J. Engineering Sciences*, 33(4), 469-483.

Lai, Y.G. (1997). "An unstructured grid method for a pressure-based flow and heat transfer solver," *Numerical Heat Transfer, Part.B*, 32, 267-281.

Lai, Y.G. (2000). "Unstructured grid arbitrarily shaped element method for fluid flow simulation," *AIAA Journal*, 38(12), 2246-2252.

Lai, Y.G., Weber, L.J., and Patel, V.C. (2003a). "Non-hydrostatic three-dimensional method for hydraulic flow simulation - Part I: formulation and verification," *ASCE J. Hydraulic Engineering*, 129(3), 196-205.

Lai, Y.G., Weber, L.J. and Patel, V.C. (2003b). "Non-hydrostatic three-dimensional method for hydraulic flow simulation – Part II: validation and application," *ASCE J. Hydraulic Engineering.*, 129(3), 206-214.

Lai, Y.G. and Yang, C.T. (2004). "Development of a numerical model to predict erosion and sediment delivery to river systems, progress report No.2: sub-model development and an expanded review," Department of the Interior, Bureau of Reclamation, Technical Service Center, Denver, Colorado.

Lai, Y.G., (2005). "River and watershed modeling: current effort and future direction", *US-China Workshop on Advanced Computational Modeling in Hydroscience & Engineering,* September19-21, 2005, Oxford, Mississippi, USA.

Lai, Y.G., (2006), "Watershed erosion and sediment transport simulation with an enhanced distributed model," *3rd Federal Interagency Hydrological Modeling Conference,* Reno, NV, April 2-6, 2006.

Lai, Y.G, Holburn, E.R., and Bauer, T.R. (2006). "Analysis of sediment transport following removal of the Sandy River Delta Dam," Project Final Report, Technical Service Center, Bureau of Reclamation, Denver, CO.

Lai, Y.G., (2009). "Watershed Runoff and Erosion Modeling with a Hybrid Mesh Model." J. Hydrological Engineering, ASCE, vol.14(1), 15-26.

Lai, Y.G., (2010). "Two-Dimensional Depth-Averaged Flow Modeling with an Unstructured Hybrid Mesh." J. Hydraulic Engineering, ASCE, vol.136(1), 12-23.

MacDonald, I., (1996). "Analysis and computation of steady open channel flow," Ph.D. Thesis, Dept. of Mathematics, Univ. of Reading, U.K.

Mihn Duc, B., Wenka, T., and Rodi, E. (1996). "Depth-averaged numerical modeling of flow in curved open channels." *Proc., 11th Conf. Computational Methods*, Cancun, Mexico (CD-ROM).

Patankar, S.V. (1980). *Numerical Heat Transfer and Fluid Flow,* McGraw-Hill, New York.

Peric, M., Kessler, R., and Scheuerer, G. (1988). "Comparison of finite volume numerical methods with staggered and collocated Grids," *Computers in Fluids,* 16(4), 389-403.

Rastogi, A., and Rodi, W. (1978). "Prediction of heat and mass transfer in open channels," *J. Hydraulic Division, ASCE,* 104(3), 397-420.

Rhie, C.M. and Chow, W.L. (1983). "Numerical study of the turbulent flow past an airfoil with trailing edge separation," *AIAA Journal*, 21(11), 1526-1532.

Rodi, W. (1980). "Turbulence models and their applications in hydraulics," Monograph, IAHR, Delft, The Netherlands.

Rodi, W. (1993). "Turbulence models and their application in hydraulics," 3rd Ed., IAHR Monograph, Balkema, Rotterdam, The Netherlands.

Shettar, A.S., and Murthy, K.K. (1996). “A numerical study of division of flow in open channels,” *J. Hydraulic Research*, 34(5), 651-675.

US Army Corps of Engineers, (1996), “Users Guide to RMA2 - Version 4.3,” US Army Corps of Engineers, Waterway Experiment Station - Hydraulic laboratory, Vicksburg, MS.

Van Rijn, L.C. (1987). “Mathematical modeling of morphological processes in the case of suspended sediment transport,” Delft Hydraulics Communication No.382.

Wu, W. and Siddle, R.C. (1995). “A distributed slope stability model for steep forested basins,” *Water Resources Research*, 31(8): 2097-2110.

Yang, C.T., Lai, Y.G., Randle, T.J., and Dario, J.A. (2003), “Development of a numerical model to predict erosion and sediment delivery to river systems: Progress Report No.1: review and evaluation of erosion models and description of SRH-2D approach,” Department of the Interior, Bureau of Reclamation Technical Service Center, Denver, Colorado.

APPENDIX A

ON MESH GENERATION USING SMS

This appendix describes how to generate a mesh using SMS. It is not the intent of this Appendix to train a user to use SMS; for such a purpose a user should resort to SMS training classes. This Appendix will focus on how to interface between SMS and SRH-2D. Note that only a portion of the SMS capabilities are used by SRH-2D. The modules used include the Map Module, Mesh Module, and Scatter Module.

A typical sequence of mesh generation procedures using SMS is as follows:

(1) Upon entering SMS, the SRH-2D-SMS template file should be loaded into SMS if Full-Interface mode is selected. Under both modes, a user needs to ensure that the model COVERAGE is set to **GENERIC 2D MESH** in the MAP module and Feature Objects/Coverage option. Upon completion of mesh generation, the mesh is stored in the 2D Generic Mesh format (2DM file). For example, casename.2DM file stores the mesh information and is used by the SRH-2D as the mesh input.

(2) The SCATTER module is used to represent the topography of the simulation solution domain. An ASCII data file (e.g., Excel files), which contains all survey points (Easting, Northing, and Bed Elevation), may be read into SMS Scatter Module. Or, topographic contour lines stored in formats such as DXF may be read into SMS Map Module and DXF may be converted into scatter points to define the topography using “DXF → Scatter Points” option. With newer SMS versions, the GIS shape file or AutoCAD files may also be imported into SMS and converted to SCATTER module. The topography information contained in the SCATTER module is used later to obtain bed elevation at mesh points through interpolation. Ideally, the survey points should cover the entire solution domain; otherwise, extrapolation should be performed within SMS.

(3) The first step in mesh generation is to use the Map Module to create the boundaries of the solution domain. Boundaries are represented with Feature Objects (nodes and arcs) with the Map Module. The topography data contained in the Scatter Module or aerial photos may be used to sketch out the solution domain. The size and location of the boundaries may be determined by factors such as the interested simulation area, the largest discharge to be simulated, etc. If possible, one solution domain is used for all possible discharges under the same topography. SRH-2D determines the wet and dry areas automatically and a larger domain may be used if an inundation extent is unknown.

(4) Once the solution domain is created, the next step is to divide the solution domain into polygons using the feature objects (nodes and arcs). Polygons are

automatically generated within Map Module using the “Feature Objects/Build Polygons” option, once all feature arcs are generated and completed. Note that the polygon creation step is very important in several ways. Firstly, each polygon may be meshed independently within the Mesh Module. Thus it may be used as a way to distribute the mesh density. For example, the main channel may be represented by a polygon so that a quadrilateral mesh is generated (with PATCH in SMS) and more mesh points may be used in the polygon. In the floodplain areas, however, polygons may be meshed with triangles (with PAVING in SMS) and much fewer points may be used. A sample mesh is shown in Figure A1 to illustrate the mesh distribution. Secondly, a polygon may be assigned a material type and the material type is used by SRH-2D to represent bed properties such as the Manning’s roughness coefficient. This way, different polygons may be used to represent spatial distributions of bed roughness.

(5) Once all polygons are generated, a mesh may be generated and a material type is assigned for each polygon. A pop-up window will appear to carry out the task by clicking the polygon within the Map Module. Several mesh types are available with SMS, and the most useful ones are the PATCH and PAVING. PATCH creates a quadrilateral structured mesh and works on four sided polygons only while PAVING creates a triangular unstructured mesh and works on any polygons. It is recommended that the main channel or special areas (e.g., structures and levees) be meshed with PATCH and the remaining areas be meshed with PAVING. The mesh density and distribution may be changed and the polygon/material type may be assigned within the pop-up window. A user should consult the SMS manual for more details on mesh generation. Do not be afraid to make mistakes as SMS allows you to revisit the mesh generation and change/modify the mesh any way necessary.

(6) Once all polygonal meshes are generated, the mesh may be assembled together by using the “Feature Objects/Map→ 2D Mesh” option within the Map Module. “Merging Triangles” option may be used to reduce the number of cells while keeping the mesh points the same. This completes the 2D mesh generation and the mesh may be displayed for examination. Steps (4) to (6) may be repeated to optimize the mesh until a satisfactory final mesh is obtained.

(7) Once the mesh is finalized, the bed elevation of each point is interpolated from the scatter data sets created in Step (2). This is accomplished by going to the Scatter Module and using the “Scatter/Interpolate-to-Mesh” option. The bed topography represented by the mesh may be examined by plotting the contour lines in the Display Option. Also, ensure that linear elements are used for the mesh (versus the Quadratic) by displaying the mesh points. If midpoints of element edges are displayed, elements are quadratic. Conversion from quadratic to linear may be carried out within the Mesh Module with the “Elements/Linear – Quadratic” option.

(8) Finally, NODESTRINGS are created within the Mesh Module. Each nodestring represents a boundary segment of the solution domain and is used by SRH-2D to specify the boundary types and boundary conditions (see the BOUNDARY SEGMENT DEFINITION command). All external boundaries of the solution domain ate setup as WALL boundaries automatically by SRH-2D. Therefore, only boundaries other than WALL boundaries need to be created using NODESTRING here. For most applications, only inlets and exits are needed. If SMS is not used as the Full-Interface to SRH-2D, a user should take a note of the nodestring order in which it is created since the order is used as the nodestring ID. For example, the first created node string has an ID of 1, and the fifth nodestring has an ID of 5, etc. The nodestring ID will be used to specify boundary conditions using the SRH-2D preprocessor. If a user forgets the nodestring IDs, the mesh file, case.2DM, may be viewed to decide the order (and ID) that is listed near the end of the file with the NS cards.

(9) The above procedures complete the mesh generation process and if the project file is saved with the name of “case”, an ASCII mesh file will be created by SMS with the name of case.2DM. This 2DM file will be used by SRH-2D.

(10) If Full-Interface mode is chosen, Chapter 3 should be consulted to see how to set up SRH-2D model parameters, boundary conditions, and Manning’s coefficients.

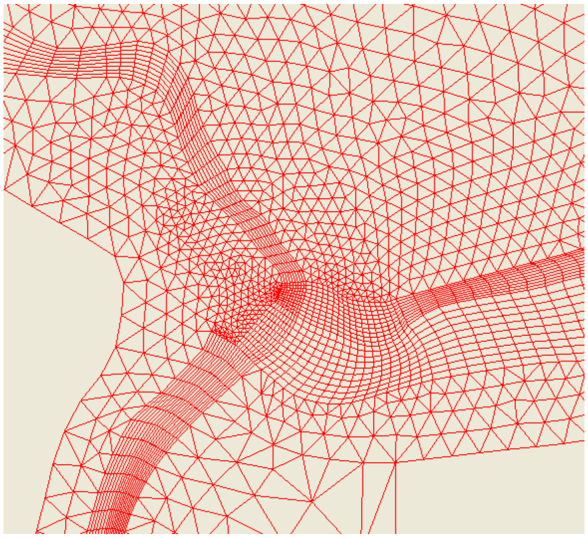


Figure A1. A Sample Mesh to Represent Main Channel and Floodplain

APPENDIX B

INPUT AND OUTPUT FORMATS

This appendix provides a description on how some of the input and output formats are used with SRH-2D.

B.1 SRH Formats

SRHN and SRHC formats are used to store the simulation results, final or intermediate, so that SMS, ArcGIS, or Excel may be used to view and process the results. Both formats are ASCII-based, spread-sheet type data which may also be imported into Excel for data manipulation. SRHC format stores all variables at the mesh (element) centers, while SRHN format stores all variables at the mesh nodal points.

B.2 TECPLOT Format

TECPLOT format is used to store the simulation results, final or intermediate, that may be imported into TECPLOT, a post-processing graphical software. Users are referred to the TECPLOT user's manual for details about the TECPLOT program.

TECPLOT program.

B.3 XMDF Format

XMDF format is a special NCSA format used by SMS to store model results.

B.4 GENERIC Format (not supported currently)

GENERIC format is an output format offered by SRH-2D as a way of obtaining the simulation results when a user does not have access to graphical packages that use the TECPLOT or SMS formats. With the GENERIC format, a user may convert the result file into other formats so that other readily available post-processing packages may be used. With the GENERIC format, an output file, casename_GNR.dat, is created.

The GENERIC format file is created with the following FORTRAN statement:

```
write(*,*) FILE-DESCRIPTION        !one record of text
write(*,*) Nvar                    !number of dependent variables
```

```
write(*,*) Variable-List                !list of variable names in the file

write(*,*) Nnode,Nelem                  !number of nodes & elements

DO Ivar=1,Nvar                          !loop over all variables
    Write(*,*) (Var(i,Ivar),i=1,Nnode)
ENDDO

DO I=1,Nelem                            !loop over all elements
  Write(*,*) Nnd                        !number of nodes for the element
  Write(*,*) (NodeID(j),J=1,Nnd)        !list of nodes of the element
ENDDO
```

APPENDIX C

DYNAMIC INPUT FILE (DIP)

Some of the frequently used parameters may be set up or modified during SRH-2D execution. This dynamic setup and change of solution parameters are achieved using the SRH-2D **Dynamic InPut (DIP) file**. The DIP file is a text file named casename_DIP.dat and it has the following format:

$DATAC
parameter assignment statement
$ENDC

A sample copy of the _DIP file may be obtained from the tutorial cases that are available through the SRH-2D distribution package.

A number of parameter assignment statements may be listed and each statement has the following syntax:

parameter-name = parameter-value

Available parameters which may be changed using the DIP file are listed below:

DTNEW = r_time

This is to change the flow simulation time step to ***r_time*** in second.

TOTAL_SIMULATION_TIME = tt

tt is the total simulation time in hours.

IREST = l

l equals 0 or 1 to specify whether the simulation is a new run or a restart (hot-start) run. SRH-2D sets *IREST=0*, as default. *IREST=0* means that it is a new run and the initial condition is set up using DRY or filename option; *IREST*=1 is for restart or hot-start run and the simulation would continue from a previous run stored in case_RST.dat RST file.

TIME_INTERVAL = ti

used for steady or unsteady simulation; it allows SRH-2D to write out intermediate results every ***ti*** hours. Output file name would be, e.g., case_SMSi.dat.

DAMP =r this allows a user to choose the amount of damping added to the 2nd-order discretization scheme for the convection term. 0.3 < ***r*** < 1.0 is recommended. The default value of 0.99 is used.

NITER = i i is the number of iterations within each time step. Note that i=1 is automatically set up by SRH-2D for a steady state simulation, and i=3 is automatically setup for an unsteady time accurate simulation.

RELAX_H = r specify relaxation of the continuity equation to ***r***, where ***r*** typically ranges from 0.001 to 0.5 with a smaller value for heavier relaxation. Typically, 0.1 should work for most problems.

RELAX_UV=r specify relaxation of the momentum equations to ***r***, where ***r*** typically ranges from 1.0 to 100.0 with a higher value for heavier relaxation. Typically, 1.0 works for most applications.

A_TURB = r This is to set the depth-averaged parabolic turbulent model coefficient. ***r*** ranges from 0.3 to 1.0; default is 0.7.

APPENDIX D

Format of Time Series Function and General Function

Both time series function, in the form of (time, var), and the general function, in the form of (var1, var2), may be used to specify boundary conditions. Each function is an input to SRH-2D through a data file. SRH-2D requires the use of a fixed format as discussed in this Appendix. Failure of storing the data in the format explained here will lead to errors in the modeling.

Time series function defines a data set as *var=f(time)*, where *time* in HOUR and var is discharge or stage (water surface elevation) whose unit is to be specified when the function is used as boundary conditions. A time series function is defined with a discrete set of data stored in a file. The data file is in ASCII format with two columns of data as follows:

Comments	
Comments	
Comments	
time(1)	*var*(1)
time(2)	*var*(2)
…	…
time(n)	*var(n)*

The first three rows are for comments about the dataset while the rest of the rows provide (*time*, *var*) data points (a total of *n* points).

A general function defines a data set as *var2=f(var1)*, where *var1* is the independent variable and *var2* is the dependent variable. The specific meaning of the two variables is specified by the first row (line) of the data file which has the following options: RATING_CURVE is the only option at present. They are explained below.

RATING_CURVE: If RATING_CURVE option is specified on the first row, *var1* is flow discharge and *var2* is stage (water surface elevation). This function is intended to be applied at an exit for an unsteady simulation.

A general function is defined with a discrete set of data stored in a file. The data file is in ASCII format with two columns of data as follows:

RATING_CURVE
Comments
Comments

var1(1)	*var2*(1)
var1(2)	*var2*(2)
…	…
var1(*m*)	*var2*(*m*)

The first row, RATING_CURVE, specifies the function type, and the next two rows are for comments about the dataset. Starting from the 4th row, up to *m* pairs of data (*var1*, *var2*) are given.

APPENDIX E

COMMON ERRORS

Listed below are common mistakes by users. This Appendix may be referred to if errors happen.

1. SRH-2D blows up quickly even with very small time step (<1.0 second): If SRH-2D blows up quickly even with a time step less than one second, the chance is that some errors are made by users in the input. Some possibilities: (1) The location of the inlet or exit boundary is misplaced unknowingly; (2) The water surface elevation at an exit is wrong in value or in unit leading to outrageous back flow into the solution domain; (3) The unit of mesh and/or the unit of water elevation or discharge at the boundaries are mistakenly specified; etc. Users are encouraged to edit the 2DM file directly (e.g., with Nodepad) to check and correct potential errors. Look at the bottom section of the 2DM file between START2DMBC and END2DMBC: "MAT" is Manning's coefficient; "GP" is input parameter; "BCN" is monitor point, "BCS" is boundary conditions for all nodestrings listed in the "ND" cards ended by a negative number.

2. Exterior mesh nodes are in the nodestrings for boundary conditions? Users may create nodestrings in SMS in which one or more nodal points are located inside the mesh instead of on the exterior boundary, or one or more nodes are missing in the nodestring list. For such circumstances, SRH-2D would issue an error message. Suggestion: go back to SMS, delete the problem nodestring, and recreate the nodestring (Hint: "Control" key may be used to ensure that only exterior boundary nodes are selected). Note: only inlets and exits are normally needed and walls for banks and/or water edges do not have to be included in the nodestrings as SRH-2D will set them up automatically.

3. Get a "bad mesh" error message? If SRH-2D issues an error message complaining a bad mesh, a cell (element) ID is also given. Users should use SMS (or other software) to inspect that element. One of several nodes around this cell may be bad that a bad mesh cell is formed. Use the SMS mesh editor to correct the mesh. One may also want to use SMS tool to check the mesh.

4. Is 2DM file inspected? Due to potential bugs in different versions of SMS (we have not tested all possible versions and we do find problems with some versions), it is suggested that the final 2DM file be inspected before running SRH-2D. Go to the bottom of a 2DM file using a text editor such as Notepad (after nodestring sections with card indicator "ND"). Check parameters within the

2DMBC section which is included between “BEG2DMBC” and “END2DMBC”. Manning’s coefficients are listed under “MAT” heading, global parameters are under “GP”, boundary conditions are under “BCS”, and monitor points are under “BCN”.

5. Difference in using “RST” option and IREST=1 option: Note the difference in using “RST” option to set up the initial conditions and IREST=1 option (setup in the _DIP.dat file) for restart or hot start run. “RST” option uses a RST file, as initial conditions, generated by another model run which has different flow parameters but the same mesh. Only the main dependent variables are used to set up the initial condition and the rest of the parameters such as boundary conditions are determined by the input file. IREST=1 option, on the other hand, is intended only for hot start run. It is used to continue a previously stopped run and the RST file is exactly the same problem.

APPENDIX F

SPECIAL TREATMENT IN MESH ZONES

A number of special treatments can be done to user selected zones on the river bed. Partial-Interface mode should be used to invoke these special treatments. They are discussed below.

Momentumless Water Source/Sink

This option allows a user to add to or remove from the river bed with a specified amount of water. It may be used to “simulate” a spring water, water withdraw at an intake, etc.

After entering 1, the command, NUMBER-OF-MOMENTUMLESS-SOURCE/SINK, will appear to enter the number of such sources/sinks (NSOURCE). Each source/sink refers to one area of the bed where the treatment is to be applied. Each source/sink will be defined by two parameters: (1) all mesh cells covering the bed area; and (2) the flow rate (in cfs or cms) with which water is added to or removed from the river.

For each source/sink, “SOURCE/SINK-INFORMATION” would appear to enter three parameters: NCELL QRATE UNIT. NCELL is the number of mesh cells that covers the source/sink area, QRATE is the flow rate with which water is added to or removed from the river, and UNIT is the unit of QRATE with CFS or CMS. The, “Enter: The-List-of-Mesh-Cell-IDs” command will appear so that a list of mesh cell IDs may be entered by the user. The cell ID may be found with SMS by displaying the element ID, and note that a number of rows may be used to enter all IDs. Each row is limited to 20 integers.