



# **TUTORIALS**

---

Version 6.0

*SMS 6.0*

Copyright © 1998 Brigham Young University – Environmental Modeling Research Laboratory October 27, 1998.

All Rights Reserved

Unauthorized duplication of the *SMS* software or user's manual is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING RESEARCH LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *SMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *SMS* is a product of the Environmental Modeling Research Laboratory of Brigham Young University and is distributed by contract through Environmental Modeling Systems, Inc. (EMS-I). For more information about this software and related products, contact EMS-I:

EMS-I  
719 North 1890 West Suite 38-B  
Provo, UT 84601

Phone:801-373-5200 FAX:801-375-6410  
info@ems-i.com  
<http://www.ems-i.com>

# TABLE OF CONTENTS

|          |  |            |
|----------|--|------------|
| <b>1</b> | <b>INTRODUCTION .....</b>                                      | <b>1-1</b> |
| 1.1      | REFERENCE MANUAL.....  | 1-2        |
| 1.2      | SUGGESTED ORDER OF COMPLETION .....                            | 1-2        |
| 1.3      | MODULES.....   | 1-3        |
| 1.4      | DEMO VS. WORKING MODES .....                                   | 1-3        |
| <b>2</b> | <b>OVERVIEW OF SMS.....</b>                                    | <b>2-1</b> |
| 2.1      | INTRODUCTION.....  | 2-1        |
| 2.2      | GETTING STARTED .....  | 2-1        |
| 2.3      | THE SMS SCREEN.....  | 2-2        |
| 2.3.1    | <i>The Main Graphics Window</i> .....                          | 2-2        |
| 2.3.2    | <i>The Toolbox</i> .....                                       | 2-2        |
| 2.3.3    | <i>The Edit Window</i> .....                                   | 2-3        |
| 2.3.4    | <i>The Menu Bar</i> .....                                      | 2-3        |
| 2.4      | USING A BACKGROUND IMAGE.....                                  | 2-3        |
| 2.4.1    | <i>Importing The Image</i> .....                               | 2-3        |
| 2.4.2    | <i>Registering The Image</i> .....                             | 2-3        |
| 2.4.3    | <i>Resampling The Image</i> .....                              | 2-4        |
| 2.5      | USING FEATURE OBJECTS .....                                    | 2-5        |
| 2.6      | CREATING FEATURE ARCS .....                                    | 2-6        |
| 2.7      | MANIPULATING COVERAGES.....                                    | 2-8        |
| 2.8      | REDISTRIBUTING VERTICES.....                                   | 2-8        |
| 2.9      | DEFINING POLYGONS .....  | 2-9        |
| 2.10     | ASSIGNING MESHING PARAMETERS.....                              | 2-10       |
| 2.10.1   | <i>Creating a Refine Point For Adaptive Tessellation</i> ..... | 2-10       |
| 2.10.2   | <i>Defining a Coons Patch</i> .....                            | 2-11       |
| 2.10.3   | <i>Removing Drawing Objects</i> .....                          | 2-12       |
| 2.11     | APPLYING BOUNDARY CONDITIONS.....                              | 2-13       |
| 2.11.1   | <i>Defining Arc Groups</i> .....                               | 2-13       |
| 2.11.2   | <i>Assigning The Boundary Conditions</i> .....                 | 2-14       |
| 2.12     | ASSIGNING MATERIALS TO POLYGONS .....                          | 2-15       |
| 2.12.1   | <i>Displaying Material Types</i> .....                         | 2-15       |
| 2.13     | CONVERTING FEATURE OBJECTS TO A MESH .....                     | 2-16       |
| 2.14     | EDITING THE FEATURE OBJECT MESH .....                          | 2-17       |
| 2.15     | INTERPOLATING TO THE MESH.....                                 | 2-17       |
| 2.16     | RENUMBERING THE MESH .....                                     | 2-18       |
| 2.17     | SAVING A SUPERFILE .....                                       | 2-18       |
| 2.17.1   | <i>Saving Model Simulation Files</i> .....                     | 2-19       |
| 2.18     | CONCLUSION.....  | 2-19       |
| <b>3</b> | <b>MESH EDITING .....</b>                                      | <b>3-1</b> |
| 3.1      | IMPORTING TOPOGRAPHIC DATA.....                                | 3-1        |
| 3.2      | TRIANGULATING THE NODES.....                                   | 3-2        |
| 3.3      | DELETING OUTER ELEMENTS .....                                  | 3-3        |
| 3.4      | DELETING THIN TRIANGLES .....                                  | 3-3        |

|          |   |            |
|----------|---|------------|
| 3.5      | MERGING TRIANGLES .....                               | 3-4        |
| 3.6      | EDITING INDIVIDUAL ELEMENTS .....                     | 3-5        |
| 3.6.1    | Using the Split / Merge Tool .....                    | 3-6        |
| 3.6.2    | Using the Swap Edge tool .....                        | 3-7        |
| 3.7      | SMOOTHING THE BOUNDARY .....                          | 3-9        |
| 3.8      | RENUMBERING THE MESH .....                            | 3-11       |
| 3.9      | CHANGING THE CONTOUR OPTIONS .....                    | 3-12       |
| 3.10     | CHECKING THE MESH QUALITY .....                       | 3-13       |
| 3.11     | SAVING THE MESH .....                                 | 3-16       |
| 3.12     | CREATING RECTANGULAR PATCHES .....                    | 3-16       |
| 3.12.1   | Gathering Cross Section Data .....                    | 3-16       |
| 3.12.2   | Creating the Mesh with Rectangular Patches .....      | 3-19       |
| 3.12.3   | A Note on Rectangular Patches .....                   | 3-21       |
| 3.13     | REFINING ELEMENTS .....                               | 3-21       |
| 3.13.1   | Inserting Breaklines .....                            | 3-22       |
| 3.13.2   | Using the Refine Command .....                        | 3-24       |
| 3.14     | FINISHING THE MESH .....                              | 3-24       |
| 3.15     | CONCLUSION .....                                      | 3-25       |
| <b>4</b> | <b>BASIC RMA2 ANALYSIS .....</b>                      | <b>4-1</b> |
| 4.1      | INTRODUCTION .....                                    | 4-1        |
| 4.2      | DEFINING MATERIAL PROPERTIES .....                    | 4-2        |
| 4.3      | CHECKING THE MODEL .....                              | 4-3        |
| 4.4      | SAVING THE SIMULATION .....                           | 4-3        |
| 4.5      | USING GFGEN .....                                     | 4-3        |
| 4.6      | USING RMA2 .....                                      | 4-4        |
| 4.7      | CONCLUSION .....                                      | 4-4        |
| <b>5</b> | <b>BASIC FESWMS ANALYSIS .....</b>                    | <b>5-1</b> |
| 5.1      | INTRODUCTION .....                                    | 5-1        |
| 5.2      | CONVERTING ELEMENTS .....                             | 5-2        |
| 5.3      | DEFINING MATERIAL PROPERTIES .....                    | 5-2        |
| 5.4      | SETTING MODEL CONTROLS .....                          | 5-3        |
| 5.5      | MODEL CHECK .....                                     | 5-4        |
| 5.6      | SAVING THE SIMULATION .....                           | 5-5        |
| 5.7      | USING FLO2DH .....                                    | 5-5        |
| 5.8      | CONCLUSION .....                                      | 5-5        |
| <b>6</b> | <b>2D POST PROCESSING .....</b>                       | <b>6-1</b> |
| 6.1      | INTRODUCTION .....                                    | 6-1        |
| 6.2      | DATA SETS .....                                       | 6-1        |
| 6.3      | USING THE DATA BROWSER .....                          | 6-2        |
| 6.4      | CREATING NEW DATA SETS WITH THE DATA CALCULATOR ..... | 6-3        |
| 6.5      | CREATING ANIMATIONS .....                             | 6-4        |
| 6.5.1    | Creating a Film Loop Animation .....                  | 6-4        |
| 6.5.2    | Creating a Flow Trace Animation .....                 | 6-5        |
| 6.6      | PLOTS .....   | 6-6        |
| 6.7      | CONCLUSION .....                                      | 6-7        |
| <b>7</b> | <b>ADVANCED RMA2 ANALYSIS .....</b>                   | <b>7-1</b> |
| 7.1      | INTRODUCTION .....                                    | 7-1        |
| 7.2      | DEFINING MATERIAL PROPERTIES .....                    | 7-2        |
| 7.3      | CREATING NODESTRINGS .....                            | 7-2        |

|           |  |             |
|-----------|--|-------------|
| 7.4       | DEFINING A DYNAMIC SIMULATION .....                              | 7-3         |
| 7.5       | DEFINING TRANSIENT BOUNDARY CONDITIONS .....                     | 7-4         |
| 7.5.1     | <i>Defining Flow Boundary Conditions</i> .....                   | 7-5         |
| 7.5.2     | <i>Defining Head Boundary Conditions</i> .....                   | 7-5         |
| 7.6       | RUNNING THE MODEL CHECKER .....                                  | 7-6         |
| 7.7       | SAVING THE SIMULATION .....                                      | 7-6         |
| 7.8       | USING REVISION RECORDS .....                                     | 7-6         |
| 7.9       | RUNNING THE INITIAL SIMULATION .....                             | 7-7         |
| 7.9.1     | <i>Running GFGEN</i> .....                                       | 7-7         |
| 7.9.2     | <i>Running RMA2</i> .....  | 7-7         |
| 7.10      | DEFINING A STEADY STATE SIMULATION .....                         | 7-8         |
| 7.11      | DEFINING CONSTANT BOUNDARY CONDITIONS .....                      | 7-8         |
| 7.12      | SAVING THE NEW SIMULATION .....                                  | 7-9         |
| 7.13      | RUNNING THE FINAL SIMULATION .....                               | 7-9         |
| 7.14      | CONCLUSION .....   | 7-9         |
| <b>8</b>  | <b>ADVANCED FESWMS ANALYSIS .....</b>                            | <b>8-1</b>  |
| 8.1       | INTRODUCTION .....   | 8-1         |
| 8.2       | OPENING THE GEOMETRY .....                                       | 8-1         |
| 8.3       | DEFINING MATERIAL PROPERTIES .....                               | 8-2         |
| 8.4       | CREATING THE HOTSTART FILE .....                                 | 8-3         |
| 8.4.1     | <i>Assigning Boundary Conditions</i> .....                       | 8-3         |
| 8.4.2     | <i>Creating Weirs</i> .....                                      | 8-5         |
| 8.4.3     | <i>Saving The Data</i> .....                                     | 8-7         |
| 8.4.4     | <i>Using FLO2DH</i> .....  | 8-8         |
| 8.5       | REWORKING THE SOLUTION .....                                     | 8-8         |
| 8.5.1     | <i>Changing The Boundary Conditions</i> .....                    | 8-8         |
| 8.5.2     | <i>Editing The Weir Data</i> .....                               | 8-9         |
| 8.5.3     | <i>Using The Hot Start file</i> .....                            | 8-9         |
| 8.5.4     | <i>Computing a New Solution File Using a Hotstart File</i> ..... | 8-9         |
| 8.6       | CONCLUSION .....   | 8-10        |
| <b>9</b>  | <b>SED2D-WES ANALYSIS .....</b>                                  | <b>9-1</b>  |
| <b>10</b> | <b>RMA4 ANALYSIS .....</b>                                       | <b>10-1</b> |
| 10.1      | INTRODUCTION .....   | 10-1        |
| 10.2      | PREPARING FOR RMA4 .....   | 10-1        |
| 10.3      | CREATING A RMA4 INPUT FILE .....                                 | 10-2        |
| 10.3.1    | <i>Creating a Constant Point Load Input File</i> .....           | 10-2        |
| 10.3.2    | <i>Creating a Variable Point Load Input File</i> .....           | 10-5        |
| 10.3.3    | <i>Creating a Salinity Intrusion Input File</i> .....            | 10-5        |
| 10.4      | USING RMA4 .....   | 10-6        |
| 10.5      | CONCLUSION .....   | 10-6        |
| <b>11</b> | <b>HIVEL ANALYSIS .....</b>                                      | <b>11-1</b> |
| 11.1      | INTRODUCTION .....   | 11-1        |
| 11.2      | CREATING MATERIALS .....   | 11-2        |
| 11.3      | CREATING NODESTRINGS .....                                       | 11-2        |
| 11.4      | DEFINING BOUNDARY CONDITIONS .....                               | 11-3        |
| 11.4.1    | <i>General Parameters</i> .....                                  | 11-3        |
| 11.4.2    | <i>Defining Steady State Flow and Head</i> .....                 | 11-4        |
| 11.4.3    | <i>Creating the Hotstart File</i> .....                          | 11-4        |
| 11.5      | SAVING THE SIMULATION .....                                      | 11-5        |

|           |   |             |
|-----------|---|-------------|
| 11.6      | USING HIVEL2D.....                              | 11-6        |
| 11.7      | CONCLUSION.....                                 | 11-6        |
| <b>12</b> | <b>CGWAVE ANALYSIS .....</b>                    | <b>12-1</b> |
| 12.1      | INTRODUCTION .....                              | 12-1        |
| 12.2      | OPENING THE DATA.....                           | 12-1        |
| 12.3      | CREATING A WAVELENGTH FUNCTION .....            | 12-2        |
| 12.4      | CREATING A SIZE FUNCTION .....                  | 12-3        |
| 12.5      | CREATING A BACKGROUND SCATTER SET .....         | 12-3        |
| 12.6      | DEFINING THE DOMAIN .....                       | 12-4        |
| 12.6.1    | <i>Creating the Coastline.....</i>              | 12-4        |
| 12.6.2    | <i>Creating the domain.....</i>                 | 12-4        |
| 12.7      | CREATING THE FINITE ELEMENT MESH .....          | 12-5        |
| 12.7.1    | <i>Setting Up The Polygon.....</i>              | 12-5        |
| 12.7.2    | <i>Generating The Elements .....</i>            | 12-6        |
| 12.8      | MODEL CONTROL .....                             | 12-7        |
| 12.9      | SAVING THE CGWAVE DATA.....                     | 12-8        |
| 12.10     | RUNNING CGWAVE .....                            | 12-8        |
| 12.11     | CONCLUSION.....                                 | 12-8        |
| <b>13</b> | <b>ADCIRC ANALYSIS .....</b>                    | <b>13-1</b> |
| 13.1      | INTRODUCTION .....                              | 13-1        |
| 13.2      | CREATING THE MESH.....                          | 13-1        |
| 13.2.1    | <i>Creating The Domain.....</i>                 | 13-2        |
| 13.2.2    | <i>Defining Mesh Generation Parameters.....</i> | 13-3        |
| 13.2.3    | <i>Generating The Finite Element Mesh.....</i>  | 13-4        |
| 13.3      | SETTING MODEL PARAMETERS .....                  | 13-5        |
| 13.4      | SAVING THE SIMULATION .....                     | 13-5        |
| 13.5      | RUNNING ADCIRC.....                             | 13-5        |
| 13.6      | VIEWING SOLUTION FILES.....                     | 13-6        |
| 13.7      | CONCLUSION.....                                 | 13-6        |
| <b>14</b> | <b>BASIC WSPRO ANALYSIS.....</b>                | <b>14-1</b> |
| 14.1      | INTRODUCTION .....                              | 14-1        |
| 14.2      | SETTING UP SMS FOR WSPRO .....                  | 14-1        |
| 14.3      | DEFINING A CROSS SECTION .....                  | 14-2        |
| 14.3.1    | <i>Creating The Section.....</i>                | 14-2        |
| 14.3.2    | <i>Importing Geometry Points .....</i>          | 14-2        |
| 14.3.3    | <i>Assigning Section Parameters.....</i>        | 14-3        |
| 14.4      | INTERPOLATING CROSS SECTIONS.....               | 14-3        |
| 14.5      | DEFINING MATERIAL PROPERTIES.....               | 14-4        |
| 14.5.1    | <i>Creating Section Break Points .....</i>      | 14-4        |
| 14.5.2    | <i>Creating Materials .....</i>                 | 14-5        |
| 14.5.3    | <i>Defining Material Roughness.....</i>         | 14-6        |
| 14.5.4    | <i>Assigning Materials To a Section.....</i>    | 14-6        |
| 14.6      | MODEL PARAMETERS .....                          | 14-7        |
| 14.7      | SAVING A SIMULATION.....                        | 14-7        |
| 14.8      | RUNNING THE WSPRO SIMULATION.....               | 14-7        |
| 14.9      | VIEWING THE PROFILE.....                        | 14-8        |
| 14.9.1    | <i>Viewing The .LST File.....</i>               | 14-8        |
| 14.9.2    | <i>Viewing The .PLT File.....</i>               | 14-8        |
| 14.10     | ADDING A BRIDGE.....                            | 14-9        |
| 14.11     | CHECKING BACKWATER AND DECK CLEARANCE .....     | 14-11       |

|           |  |             |
|-----------|--|-------------|
| 14.11.1   | Using Visual Analysis.....                 | 14-11       |
| 14.11.2   | Using the .lst file.....                   | 14-12       |
| 14.12     | CHANGING MODEL PARAMETERS .....            | 14-12       |
| 14.13     | FINISHING THE DESIGN .....                 | 14-13       |
| 14.14     | CONCLUSION.....                            | 14-13       |
| <b>15</b> | <b>ADVANCED WSPRO ANALYSIS.....</b>        | <b>15-1</b> |
| 15.1      | INTRODUCTION .....                         | 15-1        |
| 15.2      | READING A TIFF IMAGE.....                  | 15-1        |
| 15.3      | CREATING A CONCEPTUAL MODEL .....          | 15-2        |
| 15.3.1    | Defining a Centerline.....                 | 15-3        |
| 15.3.2    | Defining Cross Sections .....              | 15-4        |
| 15.3.3    | Creating an Area Property Coverage .....   | 15-5        |
| 15.3.4    | Assigning Material Types.....              | 15-6        |
| 15.3.5    | Importing Topographic Data .....           | 15-7        |
| 15.4      | SAVING THE DATA .....                      | 15-8        |
| 15.5      | CONVERTING THE CONCEPTUAL MODEL.....       | 15-8        |
| 15.6      | DEFINING THE WSPRO DATA.....               | 15-10       |
| 15.6.1    | Assigning Material Values .....            | 15-10       |
| 15.6.2    | Defining The Control Parameters.....       | 15-10       |
| 15.7      | SAVING THE SIMULATION .....                | 15-10       |
| 15.8      | RUNNING WSPRO.....                         | 15-11       |
| 15.9      | VIEWING THE RESULTS .....                  | 15-11       |
| 15.10     | CONCLUSION.....                            | 15-11       |
| <b>16</b> | <b>OBSERVATION COVERAGE.....</b>           | <b>16-1</b> |
| 16.1      | INTRODUCTION .....                         | 16-1        |
| 16.2      | OPENING THE DATA .....                     | 16-1        |
| 16.3      | VIEWING SOLUTION DATA .....                | 16-2        |
| 16.4      | CREATING AN OBSERVATION COVERAGE.....      | 16-3        |
| 16.5      | CREATING AN OBSERVATION POINT.....         | 16-3        |
| 16.5.1    | Using The Calibration Target.....          | 16-5        |
| 16.6      | READING A SET OF OBSERVATION POINTS .....  | 16-5        |
| 16.7      | GENERATING ERROR PLOTS.....                | 16-6        |
| 16.7.1    | Using The Computed vs. Observed Plot.....  | 16-7        |
| 16.7.2    | Using The Error Summary Plot.....          | 16-7        |
| 16.8      | CALIBRATING THE MODEL.....                 | 16-7        |
| 16.8.1    | Editing The Material Properties .....      | 16-7        |
| 16.8.2    | Computing a New Solution.....              | 16-8        |
| 16.8.3    | Reading The New Solution .....             | 16-8        |
| 16.8.4    | Fine-tuning the model .....                | 16-8        |
| 16.9      | USING THE ERROR VS. SIMULATION PLOT .....  | 16-9        |
| 16.10     | GENERATING OBSERVATION PROFILE PLOTS ..... | 16-9        |
| 16.11     | CONCLUSION.....                            | 16-11       |



---

## ***Introduction***

This document contains tutorials for the Surface-water Modeling System (*SMS*) version 6.0. Each tutorial is meant to provide training on a specific component of *SMS*. It is strongly suggested that you complete the tutorials before using *SMS* on a routine basis. In addition, training is available through Environmental Modeling Systems, Inc. (<http://ems-i.com>).

*SMS* is a pre- and post-processor for surface water modeling and analysis. It includes two-dimensional finite element, two-dimensional finite difference, three-dimensional finite element and one-dimensional step backwater modeling tools. Interfaces specifically designed to facilitate the utilization of several numerical models comprise the modules of *SMS*. Supported models include the USACE-WES supported *TABS-MD* (*GFGEN*, *RMA2*, *RMA4*, *RMA10*, *SED2D-WES*), *ADCIRC*, *CGWAVE*, *STWAVE*, and *HIVEL2D*. Comprehensive interfaces have also been developed for facilitating the use of the FHWA commissioned analysis packages *FESWMS* and *WSPRO*.

Each numerical model is designed to address a specific class of problem. Some calculate hydrodynamic data such as water surface elevations and flow velocities. Others compute wave mechanics such as wave height and direction. Still others track contaminant migration or suspended sediment concentrations. Some of the models support both steady-state and dynamic analyses, while others support only steady-state analysis. Some support supercritical flow, while others support only subcritical.

The finite element mesh, finite difference grid, or cross section entities, along with associated boundary conditions necessary for analysis, are created within *SMS* and then saved to model-specific files. These files are used as input to the hydrodynamic, wave mechanic, contaminant migration, and sediment transport analysis engines.

The numerical models create solution files that contain the water surface elevations, flow velocities, contaminant concentrations, sediment concentrations or other functional data at each node, cell, or section. *SMS* reads this data to create plots and animations.

*SMS* can also be used as a pre- and post-processor for other finite element or finite difference programs as long as the programs can read and write files in a supported format. *SMS* is well suited for the construction of large, complex meshes (up to hundreds of thousands of elements) of arbitrary shape.

Please note that in these tutorials, reference to a menu item will be as follows: *Menu | Menu-Item*. For example: *File | Quit* indicates to select the *Quit* item from the *File* menu.

## 1.1 Reference Manual

---

Accompanying the tutorial documentation is the *SMS Reference Manual*, which fully describes the *SMS* interface. It is suggested that the tutorials be completed prior to reading the reference manual. However, some tutorials refer sections in the reference manual. In these cases, the particular section of the reference manual should be read if a better understanding is needed.

## 1.2 Suggested Order of Completion

---

Most of these lessons are developed for two dimensional surface water modeling. If you want to use the *SMS* for its *River Module* and *WSPRO* interface, you may want to examine the this first tutorial and then skip to Lessons 14 and 15.

For other users of *SMS*, we recommend that you start with the Lessons 2 and 3, which describe the basic tools available in *SMS* for generating finite element meshes. From there, a variety of lessons could be completed, depending on the numerical model you wish to use.

Lessons 6 and 16 concentrate on using the post-processing capabilities of *SMS*. One of these uses the *RMA2* results while the other uses the *FESWMS* results. However, the techniques presented are identical for post-processing other numerical models. All files required to complete these lessons are provided, so it is possible to complete each, whether or not *SMS* has been licensed.

## 1.3 Modules

---

The *SMS* interface is divided into five modules. A module is provided for each of the basic data types supported by *SMS*. As you switch from one module to another, the tools in the *Toolbox* and the menus change. This division allows for focus on small portions of the interface at a time. The *Mesh*, *Grid*, and *River* modules have interfaces to certain numerical analysis models. For these modules, only one numerical model is available at a time.

## 1.4 Demo Vs. Working Modes

---

Some users do not require all modules or model interfaces provided in *SMS* which can be licensed individually. The icons for unlicensed modules and the menus for unlicensed model interfaces cannot be accessed. It is possible, however, to access all modules and model interfaces in *SMS* by running in *Demo Mode*.

If you have not licensed any part of *SMS*, it will automatically run in demo mode. On the other hand, if you do have a license for part of *SMS* and would like to experiment with an unlicensed modules, you can tell *SMS* to run in demo mode. To do this:

1. Select *File / Demo mode*.
2. Select *Yes* to the prompt, *Are you sure you want to delete everything?*

When you do this, a check mark appears next to this menu command. Demo mode in *SMS* allows access to all functions except the *Save* and *Print* commands. To get back to normal operation mode, select this menu item again. Note that if you have no registered modules, you cannot leave demo mode.



---

## Overview of SMS

---

### 2.1 Introduction

---

This tutorial describes the major components of the *SMS* interface and gives a brief introduction to the different *SMS* modules. It is suggested that this tutorial be completed before any other tutorial.

---

### 2.2 Getting Started

---

Before beginning this tutorial you should have installed *SMS* on your computer. If you have not yet installed *SMS*, please do so before continuing. Each chapter of this tutorial document demonstrates the use of a specific component of *SMS*. If you have not purchased all modules of *SMS*, or if you are evaluating the software, you should run *SMS* in *Demo Mode* to complete this tutorial (see section 1.4). When using *Demo Mode*, you will not be able to save files. For this reason, all files that you are asked to save have been included in the *output* subdirectory under the *tutorial* directory. When you are asked to save a file, you should instead open the file from this *output* directory. To start *SMS*, follow the instruction for the appropriate machine:

- *PC*. Open the *Start* menu, scroll to *Programs* and then to *SMS* and click on *SMS 6.0*.
- *UNIX*. Go to the *SMS* directory and type `sms` at the command line.

## 2.3 The SMS Screen

---

The *SMS* screen is divided into four main sections: the *Main Graphics Window*, the *Toolbox*, the *Edit Window*, and the *Menu Bar*, as shown in Figure 2-1. The layout of these windows may differ slightly from the figure if you are running the UNIX version of *SMS*. Two other windows, the *Plot Window* and the *River Window*, can be opened using the appropriate command from the *Display* menu.

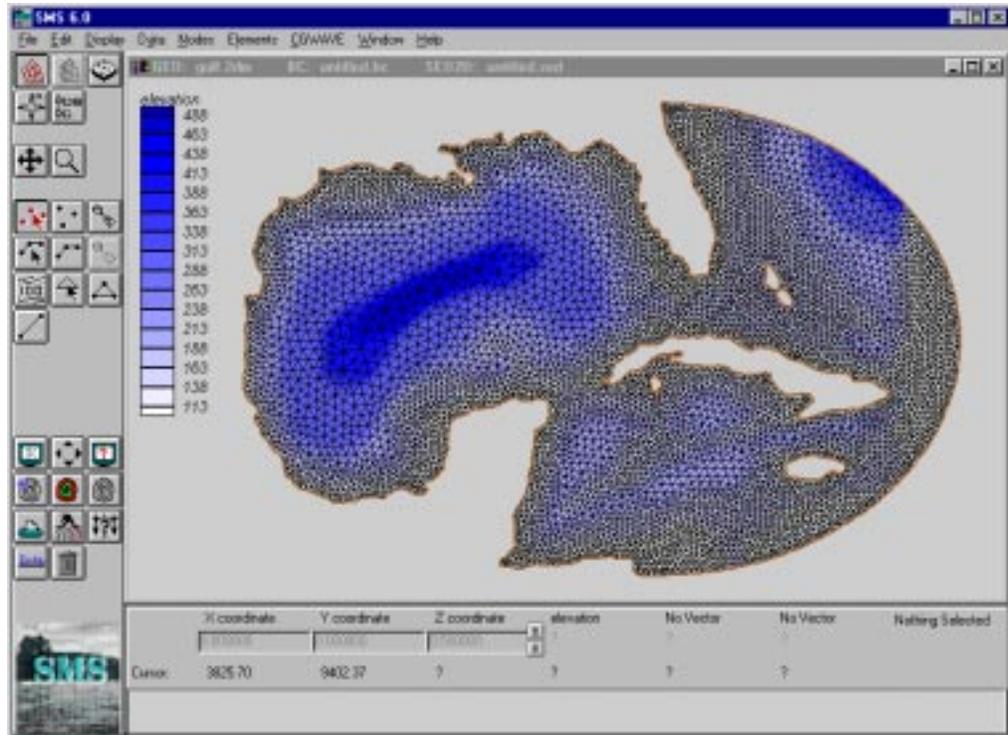


Figure 2-1. The *SMS* screen.

### 2.3.1 The Main Graphics Window

---

The *Main Graphics Window* is the biggest part of the *SMS* screen. Most of the data manipulation is done in this window. You will use it in most of the tutorial chapters.

### 2.3.2 The Toolbox

---

The *Toolbox* is broken into the following four sections:

- *Modules*. There are currently five *SMS Modules*. The function of these is described in the *SMS Reference Manual*.
- *Static Tools*. This contains a set of tools which do not change for different modules. These tools are used for manipulating the display.

- *Dynamic Tools*. These tools change according to the selected module. These tools are used for creating and editing entities specific to the module.
- *Macros*. These are shortcuts for frequently used menu commands. Not all macros are available in all modules.

### 2.3.3 The Edit Window

---

The *Edit Window* is used to change the coordinates of entities, show help messages, and display prompts for specific actions. As you move the mouse cursor around in the *Main Graphics Window*, the cursor coordinates are shown in the *Edit Window*. As you move the mouse cursor over the tools in the *Toolbox*, a help message is shown in the *Edit Window*.

### 2.3.4 The Menu Bar

---

The *Menu Bar* contains commands that are available for data manipulation. The menus shown in the *Menu Bar* depend on the module that is selected.

## 2.4 Using a Background Image

---

A good way to visualize the model is to import a digital image of the site. For this example, the image was created by scanning a portion of a USGS quadrangle map and saving the scanned image as a *TIFF* file. Once the image is imported to *SMS*, it is displayed in the background as a guide for on-screen digitizing of mesh features.

### 2.4.1 Importing The Image

---

To import the *TIFF* image:

1. Select *File | Import*.
2. In the *Import Format* dialog select the *TIF* option and click the *OK* button.
3. Open the file *stmary.tif* from the *tutorials* directory. A preview image will be shown in the *Register Image* dialog. (Do not click the *OK* button yet.)

### 2.4.2 Registering The Image

---

All *TIFF* images must be registered after being opened, except for Geo-Referenced *TIFF* images, which are automatically registered. To register a *TIFF* image, three points on the image, called *registration points*, are given real world coordinates.

When the image is drawn, it is skewed, rotated, and stretched according to the registration points. The registration points can be defined manually or can be read from a *TIFF world file*. For this example, a TIFF world file will define the registration points. To open the TIFF world file:

1. In the *Register Image* dialog click the *Import World File* button.
2. Open the file *stmary.tfw* from the *tutorial* directory. The three register points (red cross-hairs) will be assigned real world coordinates.
3. Click the *OK* button to close the *Register Image* dialog.

A progress bar appears in the *Edit Window* along with a message that the image is being resampled. After a few moments, the image will be drawn in the *Graphics Window*. The image will always be drawn under all other objects.

### 2.4.3 Resampling The Image

---

When zooming in or out, the image can become poorly sampled. It can be cleared up by resampling. To demonstrate this:

1. Choose the *Zoom*  tool from the *Toolbox*.
2. Drag a box around the main river portion, as shown in Figure 2-2. The image is somewhat blurry because of the old image sampling.
3. Switch to the *Map*  module.
4. Select *Images / Resample*. This will resample the image to the zoomed part and enhance the image resolution.



Figure 2-2 The main river portion of the image.

## 2.5 Using Feature Objects

With a background image displayed, the conceptual model can be created. A conceptual model is constructed using *feature objects* in the *Map* module. Feature objects in *SMS* include points, nodes, arcs, and polygons, as shown in Figure 2-1. Feature objects are grouped into sets called *coverages*. Each coverage represents a particular type of data, but only one coverage is active at a time.

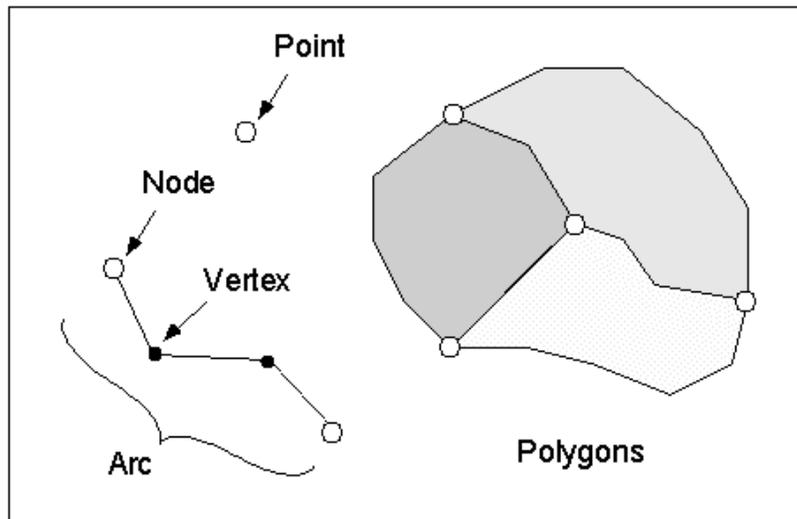


Figure 2-3 Feature Object

A *feature point* defines an  $(x, y)$  location that is not attached to an arc. Points are used to force the creation of a mesh node at a specific location. A *feature node* is the same as a feature point, except that it is attached to at least one arc.

A *feature arc* is a sequence of line segments grouped together as a polyline entity. Arcs can form polygons or represent linear features such as channel edges. The two end points of an arc are called *feature nodes* and the intermediate points are called *feature vertices*.

A *feature polygon* is defined by a closed loop of feature arcs. A feature polygon can consist of a single feature arc or multiple feature arcs, as long as a closed loop is formed.

The conceptual model in this example will consist of a single coverage, in which the river regions and the flood bank will be defined. As you go along in this tutorial you will load new coverages over the existing coverage. The new coverage will become active and the old coverage will be inactive.

## 2.6 Creating Feature Arcs

A set of feature objects can be created to show topographically important features such as a river channel and material region boundaries. Feature objects can be digitized directly inside *SMS* or they can be converted from an existing *AutoCAD DXF* file. For this example, the feature objects will be digitized inside *SMS* using the registered TIFF image as a reference. To create the feature arcs:

1. Choose the *Create Feature Arc*  tool from the *Toolbox*.
2. Click out the left river bank, as shown in Figure 2-4. As you create the arc, if you make a mistake and wish to back up, press the *BACKSPACE* key. If you wish to abort the arc and start over, press the *ESC* key. Double-click the last point to end the arc.

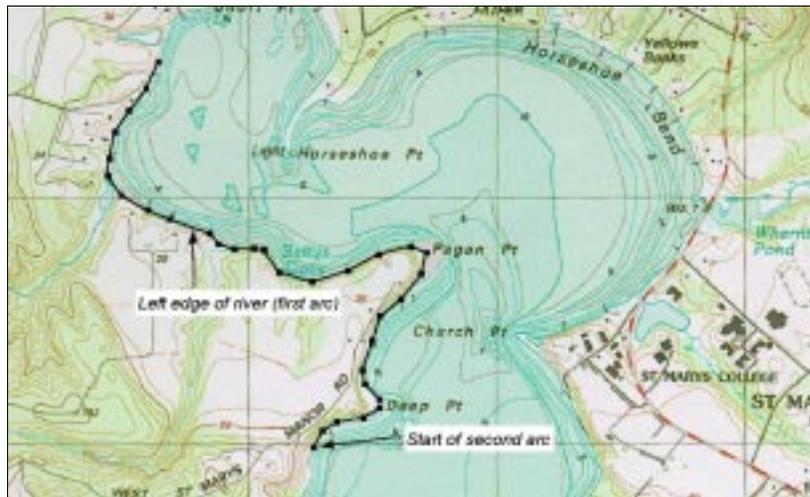


Figure 2-4 Creation of the first feature arc.

A feature arc has defined the general shape of the left river bank. Three more arcs are required to define the right river bank and the upstream and downstream river cross sections. Together, these arcs will be used to create a polygon which defines the study area. To create the remaining arcs:

- In the same manner just described, create the remaining three arcs, as shown in Figure 2-5. Remember to double-click the last point on each arc so that three separate arcs are created.

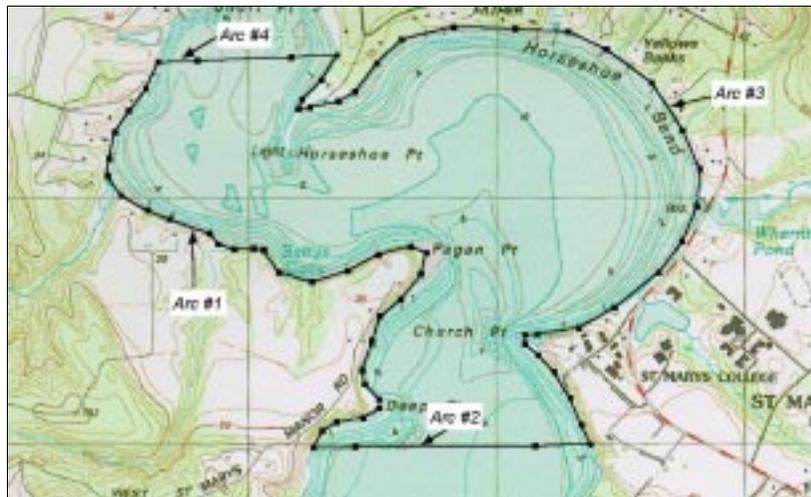


Figure 2-5 All feature arcs have been created.

You have now defined the main river channel. When creating your own models, you will proceed to create other arcs to separate material zones and define specific model features. To save time, the main features have been saved in a file. To open the file:

1. Select *File / Open*.
2. Open the file *stmary1.map* from the *tutorial* directory.

A new coverage is created from the data in the file, and the coverage you were using becomes inactive. The display should look like Figure 2-6.

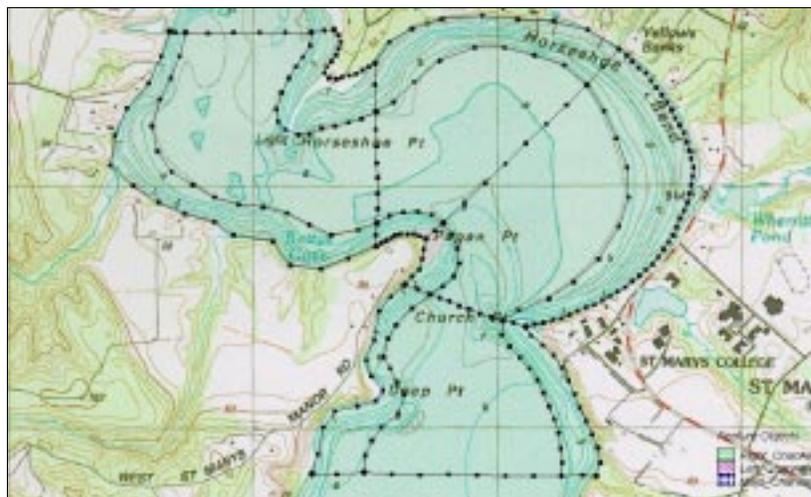


Figure 2-6 The *stmary1.map* feature object data.

## 2.7 Manipulating Coverages

---

As stated at the beginning of this tutorial, feature objects are grouped into coverages. When a set of feature objects is opened from a file, a new coverage is created. The new coverage is made active, and the previous coverage becomes inactive. Inactive coverages are drawn in a mild green color by default.

When there are many coverages being drawn, the display can become cluttered. Individual coverages can be hidden to minimize this clutter. To hide all inactive coverages:

1. Select *Feature Objects | Coverages*. The active coverage is indicated with a capital A, while any visible coverages are indicated with a lowercase v. The active coverage should be *stmary1.map*.
2. Click the *Hide All* button to hide all the coverages.
3. Highlight the *stmary1.map* coverage and turn on the *Visible* option so that it is the only visible coverage.
4. Click the *OK* button to close the *Coverages* dialog.

As this tutorial progresses, you will be asked to open a few more map files. After doing so, you might want to hide or delete the old coverages so that the display does not become too cluttered.

## 2.8 Redistributing Vertices

---

To create the feature arcs, you simply clicked out a line of points on the image without paying attention to vertex distribution. The final element density in a mesh created from feature objects matches the density of vertices along the feature arcs, so it is desirable to have a more uniform node distribution. The density vertices in a feature arc can be redistributed at a desired spacing. To redistribute vertices:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Click on the arc to the far right, labeled *Arc #1* in Figure 2-7.
3. Select *Feature Objects | Redistribute Vertices*. The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.
4. Choose the *Target spacing* option and enter a value of 470.
5. Click *OK* to redistribute the vertices along the arc.

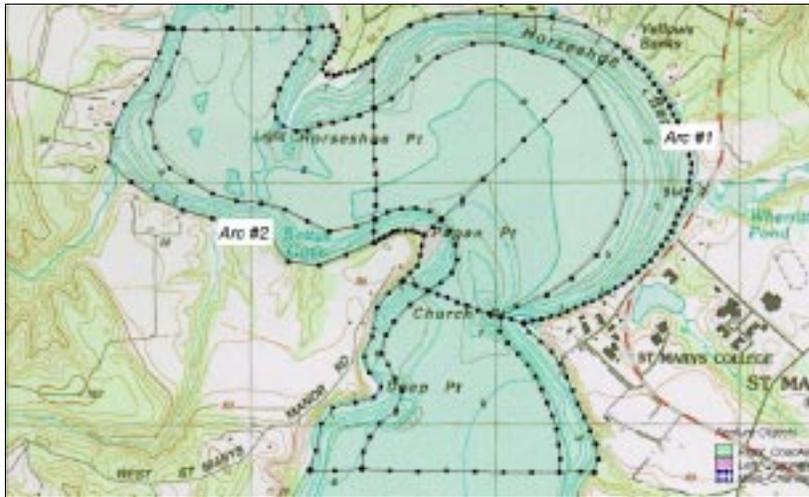


Figure 2-7 Redistribution of Vertices along arcs.

After clicking the *OK* button, the display will refresh, showing the specified vertex distribution. When you create your models, this would be done for each arc until you have the vertex spacing that you want. When you plan to use arcs in a patch, a better patch is created if opposite arcs have an equal number of vertices. In this case, you would want to use the *Number of subdivisions* option rather than the *Target spacing* option so that you can specify the exact number of vertices along each arc.

For this example, you should open another map file, which has the vertices redistributed on all the arcs. To open the map file:

- Open the file *stmary2.map*.

## 2.9 Defining Polygons

Polygons are created from a group of arcs which form a closed loop. Each polygon is used to define a specific material zone. Although polygons can be created one by one, it is faster to have *SMS* create them automatically. To have *SMS* build polygons out of the arcs:

1. Make sure no arcs are selected.
2. Select *Feature Objects | Clean* to be sure that there are no problems with the feature objects that were created. Click *OK* in the *Clean Options* dialog.
3. Select *Feature Objects | Build Polygons* and click the *OK* button at the prompt to use all feature arcs.

Although nothing appears to have changed, polygons have been built from the arcs. The polygons in this example are for defining the material zones as well as to aid in creating a better quality mesh.

## 2.10 Assigning Meshing Parameters

---

With polygons, arcs and points created, *meshing parameters* can be assigned. These meshing parameters define which automatic mesh generation method will be used to create finite elements inside the polygon. For each method, a corner node will be created at each vertex on the feature arc. The difference comes in how internal nodes are created, and how those nodes are connected to form elements.

*SMS* currently allows one of three mesh generation methods: adaptive tessellation, density meshing, and coons patches. These methods are described in detail in the *SMS Reference Manual*, so they will not be described here. Suffice it to say that adaptive tessellation is the default technique because it works for all polygon shapes, while patches require either 3 or 4 sides. Density meshing will not be performed in this example, but it is used in lessons 12 and 13.

### 2.10.1 Creating a Refine Point For Adaptive Tessellation

---

When using the default Adaptive Tessellation method, some control can be maintained over how elements are created. A *refine point* is a feature point which is created inside the boundary of a polygon and assigned a size value. When the finite element mesh is created, a corner node will be created at the location of the refine point and all element edges which touch the node will be the exact length specified by the refine point size value. To create a refine point:

1. Choose the *Select Feature Point*  tool from the *Toolbox*.
2. Double-click on the point inside the left polygon, labeled in Figure 2-8.
3. In the *Feature Point/Node Attributes* dialog, turn on the *Refine Point* option and enter a value of 100 (ft).
4. Click the *OK* button to accept the refine point.

When the finite element mesh gets generated, a mesh corner node will be created at the refine point's location, and all attached element edges will be exactly 300 feet in length. A refine point is useful when a node needs to be placed at a specific feature, such as at a high or low elevation point.

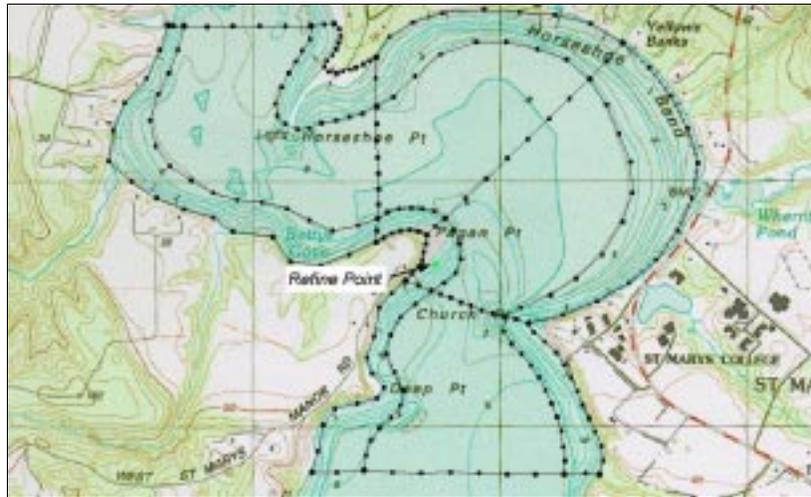


Figure 2-8 The location of the refine point.

### 2.10.2 Defining a Coons Patch

As was previously stated, the *Coons Patch* mesh generation method requires exactly three or four sides to be created. However, very rarely do exactly three or four arcs make up a polygon. Figure 2-9 shows an example of a rectangular patch made up of four sides. Notes that *Side 1* and *Side 2* are both made from multiple feature arcs.

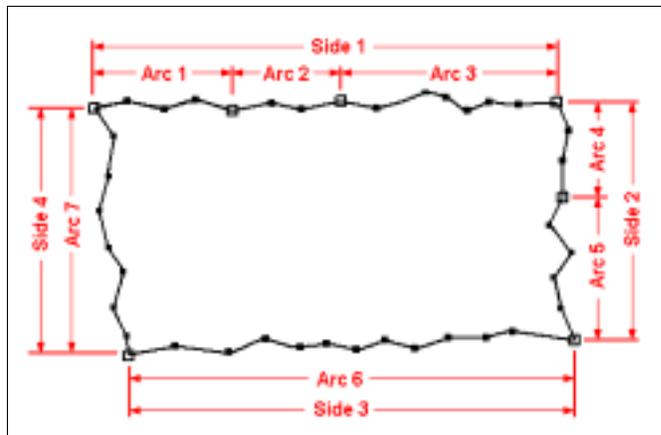


Figure 2-9 Four sides required for a rectangular patch.

SMS provides a way to define a patch from such a polygon by allowing multiple arcs to act as one. For example, the bottom middle polygon, labeled as *Patch Polygon*, contains five arcs, but it should be used to create a patch. To do this:

1. Choose the *Select Feature Polygon*  tool and double click on the bottom middle polygon.

2. In the middle of the *Feature Polygon Attributes* dialog, choose the *Select Feature Node*  tool.
3. Click on the node at the center of the left side, as seen in Figure 2-10.
4. Click the *Merge* button. This makes the two arcs on the left side be treated as a single arc. Notice that the *Patch* option is now available.
5. Select the *Patch* option. If you wish to preview the patch, click the *Preview* button.
6. Click the *OK* button to close the *Feature Polygon Attributes* dialog.

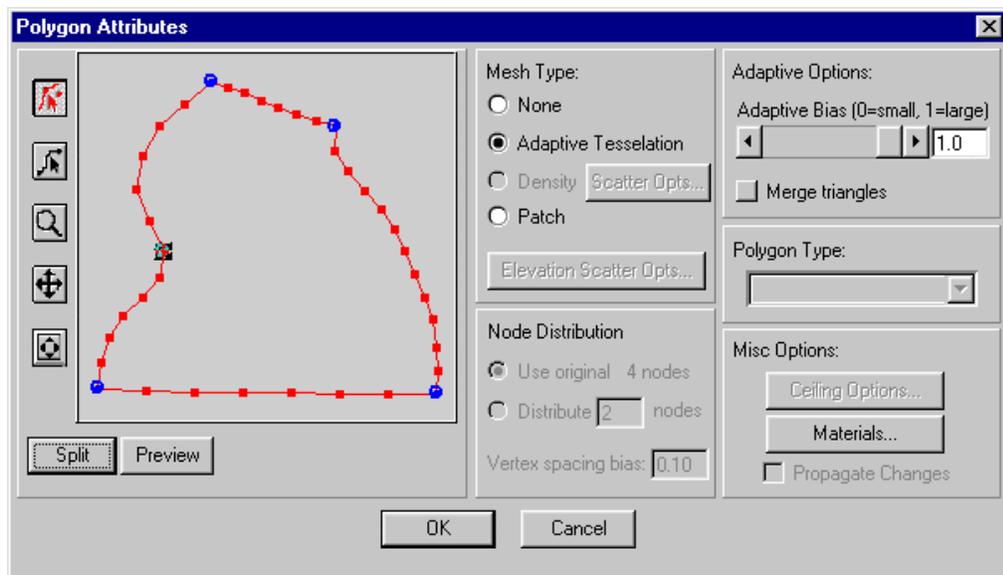


Figure 2-10 The *Feature Polygon Attributes* dialog.

When you are creating your models, you will need to set up the desired polygon attributes for each feature polygon in your model. Since this would take too much time for this tutorial, the rest of the polygons have been set up for you and saved to a map file. To import this data:

- Open the file *stmary3.map*.

In the coverage that opens, all polygon attributes have been assigned. The three main channel polygons are assigned as patches, while the other polygons are assigned as adaptive tessellation.

### 2.10.3 Removing Drawing Objects

Throughout this tutorial, *drawing objects*, such as labels and arrows, have been provided to give a description of certain feature objects. Drawing objects are not part

of a coverage, so they do not become inactive. The drawing objects that have thus far been used will not be needed anymore. To delete the drawing objects:

1. Choose the *Select Drawing Objects*  tool from the *Toolbox*.
2. Choose *Edit / Select All* to select all drawing objects.
3. Press the *DELETE* key or click the *Delete*  macro from the *Toolbox*.

## 2.11 Applying Boundary Conditions

---

Boundary conditions can be assigned to arcs, points, and for *FESWMS*, polygons. Feature arcs may be assigned a flow, head, or flux status. Feature points may be assigned velocity or head values. Feature polygons may be assigned ceiling elevation functions, but only in a *FESWMS* coverage.

The inflow for this example is across the top of the model and the outflow is across the bottom. Notice that there are three feature arcs across each of these sections. A flow rate value could be assigned to each of the arcs at the inflow. However, this would create three separate inflow nodestrings, connected end-to-end. The same situation exists at the outflow cross section.

To avoid creating three separate boundary conditions at a single cross section, an *arc group* can be defined. An arc group consists of multiple arcs that are linked together. The arc group can be assigned the boundary condition instead of assigning it at the individual arcs so that when the model is generated, only a single nodestring gets created, which spans the entire cross section.

### 2.11.1 Defining Arc Groups

---

For this example, two arcgroups will be defined, one at the inflow boundary and one at the outflow boundary. To create the arcgroups:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Holding the *SHIFT* key, select the three arcs that make up the flow cross section, labeled as *Flow Arcs* in Figure 2-11.
3. Select *Feature Objects / Create Arc Group* to create an arc group from the three selected arcs.
4. Now, select the three arcs that make up the head cross section, labeled as *Head Arcs* in Figure 2-11.
5. Select *Feature Objects / Create Arc Group*.

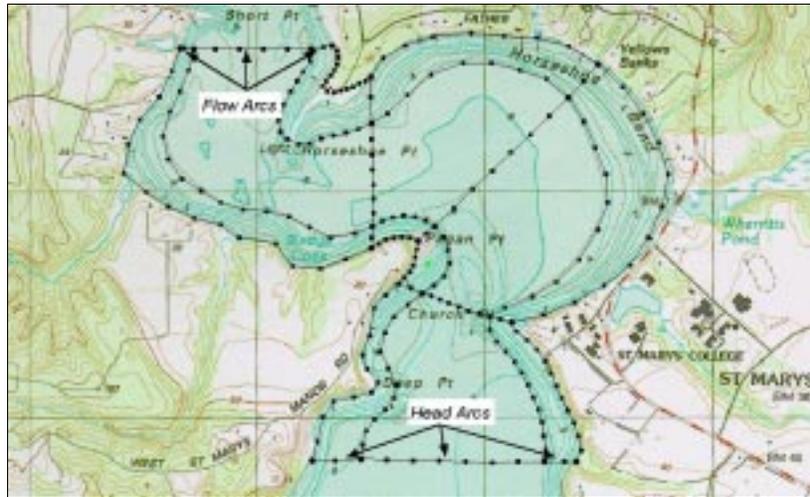


Figure 2-11 The arc groups to create.

### 2.11.2 Assigning The Boundary Conditions

With the arcgroups created, boundary conditions can now be assigned. To assign the inflow boundary condition:

1. Choose the *Select Arc Group*  tool from the *Toolbox*.
2. Double-click the arc group at the inflow cross section.
3. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button. Assign a flow of 40000 cfs.
4. Click the OK button in both dialogs.

To assign the water surface boundary condition:

1. Double-click the arc group at the outflow cross section.
2. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button. Assign a head (water surface) of 20 ft.
3. Click the OK button in both dialogs.

The inflow and outflow boundary conditions are now defined in the conceptual model. When the conceptual model is converted to a finite element mesh, SMS will create the nodestrings and assign the proper boundary conditions.

## 2.12 Assigning Materials To Polygons

Each polygon is assigned a material type. All elements generated inside the polygon are assigned the material type defined in the polygon. To assign the materials:

1. Choose the *Select Feature Polygon*  tool from the *Toolbox*.
2. Double-click on any of the polygons.
3. In the *Feature Polygon Attributes* dialog, click the *Materials* button. In the *Materials Data* dialog, highlight the correct material for the polygon, as shown in Figure 2-12, and click the *Close* button.
4. Click the *OK* button to close the *Feature Polygon Attributes* dialog.

Repeat these steps to assign the correct material type to each of the feature polygons.

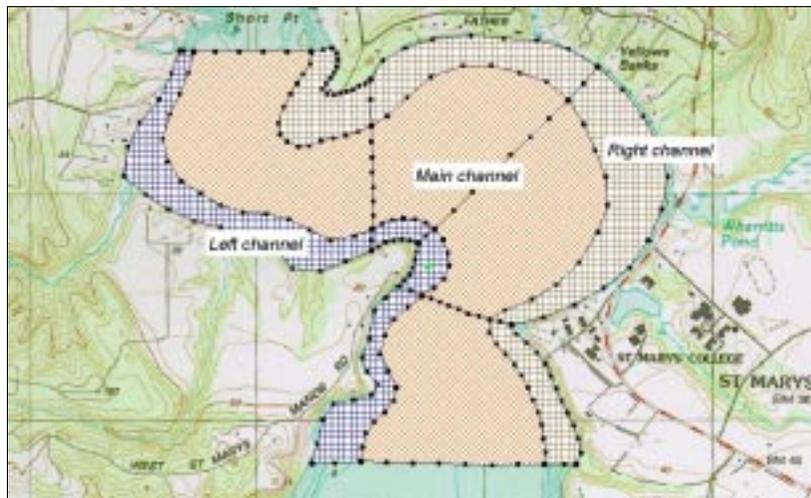


Figure 2-12 Polygons with defined material types.

### 2.12.1 Displaying Material Types

With the materials assigned to the polygons, you can fill the polygons with the material colors and patterns. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.
2. Turn on the *Polygons (Fill)* option and select the *Show Materials* option.
3. Click the *OK* button to close the *Feature Objects Display Options* dialog.

The display will refresh, filling each polygon with the material color and pattern, and a legend will appear in the lower right corner of the *Graphics Window*.

## 2.13 Converting Feature Objects To a Mesh

With the meshing techniques chosen, boundary conditions assigned, and materials assigned, we are ready to generate the finite element mesh. To do this:

1. Select *Feature Objects / Map ->2D Mesh*.
2. Make sure the *Mesh Source* is set to *Mesh From Poly*, and the *Bathymetry Source* is assigned a *Default Elevation* of 0.0.
3. Click the *OK* button to start the meshing process.

After a few moments, the display will refresh to show the finite element mesh that was generated according to the preset conditions. With the mesh created it is often desirable to delete or hide the feature arcs and the image. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.
2. Turn off the display of *Arcs*, *Nodes*, and *Polygons*.
3. Click the *OK* button to close the *Feature Objects Display Options* dialog.
4. Select *Images / Display Image* to turn the image off. The check mark will be gone from this menu item and the image will not be drawn again until it is turned back on.

The display will refresh to show the finite element mesh, as shown in Figure 2-13. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

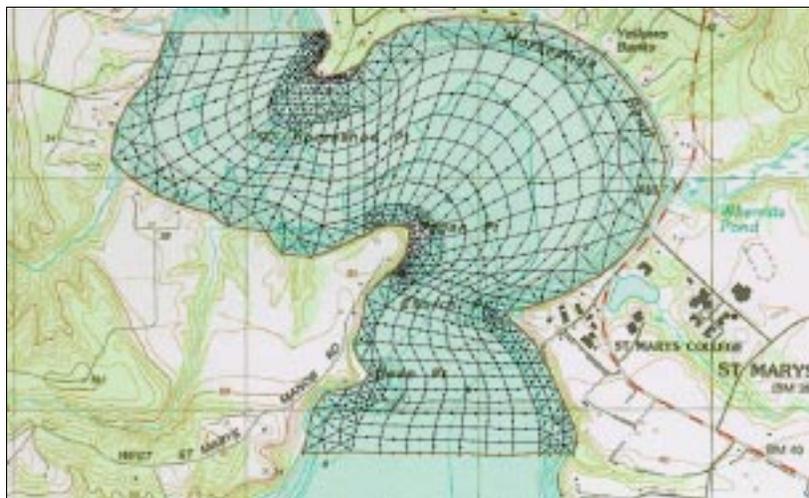


Figure 2-13 The finite element mesh that was created.

---

## 2.14 Editing the Feature Object Mesh

---

When a finite element mesh is generated by feature objects, it is not always the way you want it. An easy way to edit the mesh is to change the meshing parameters in the conceptual model, such as the distribution of vertices on feature arcs or the mesh generation parameters. Then, the mesh can be regenerated according to the new parameters. If there are only a few changes desired, they can be edited manually using tools in the mesh module. These tools are described in the *SMS Reference Manual* in the chapter on the *Mesh Module*.

When this mesh was generated, many nodestrings were created along the mesh boundary. Besides the two nodestrings that define flow and head values, the remainder of the nodestrings are unnecessary. Deleting these extra nodestrings will help eliminate problems that may arise when the mesh is run through a model. For example, GFGEN will not run if there are more than 20 nodestrings. To delete the extra nodestrings:

1. Select the *Select Nodestring*  tool.
2. Select all of the nodestrings except for the two nodestrings where the boundary conditions are defined by holding the SHIFT key while clicking in the box of each nodestring.
3. Select *Edit / Delete* or press the DELETE key.

---

## 2.15 Interpolating To The Mesh

---

The finite element mesh generated from the feature objects defines only the *X*- and *Y*- coordinates for the nodes. To get the bathymetric information, survey data saved as scatter points can be interpolated onto the finite element mesh. To open the scattered data:

4. Switch to the *Scatter Point*  module.
5. Select *File / Open* and open the file *stmaryscat.sup*.

The screen will refresh, showing a set of red crosses. Each of these points represents the location where a survey measurement was taken. There are various methods for interpolating a set of scatter points onto a finite element mesh. For more information on the interpolation methods supported by *SMS*, see the *SMS Reference Manual*. To interpolate the scattered data onto the mesh:

1. Select *Interpolation / to Mesh*.
2. In the *Interpolate To Mesh* dialog, set the *Interpolation Method* to *Linear*.

3. Turn on the *Map Elevations* option at the top left of the dialog.
4. Click the *OK* button to perform the interpolation.

The scattered data will be internally triangulated and an interpolated value is assigned to each node in the mesh. The *Map Elevations* option causes the newly interpolated value to be used as the nodal *Z*- coordinate. A status bar in the *Edit Window* shows how the interpolation is proceeding.

As with the feature objects, the scattered data will no longer be needed and may be hidden or deleted. To hide the scatter point data:

1. Click the *Display Options*  macro from the *Toolbox*.
2. Turn off the *Scatter point symbols* option.
3. Click the *OK* button to close the *Scatter Point Set Display Options* dialog.

## 2.16 Renumbering the Mesh

---

The process of creating and editing a finite element mesh can cause the node and element ordering to become disorganized. A good mesh ordering can be restored by renumbering the mesh. To do this:

1. Choose the *Create Nodestring* tool  from the *Toolbox*.
2. Select the flow nodesrting at the top of the mesh.
3. Select *Elements / Renumber*. Click the *OK* button or press the ENTER key to begin renumbering.

## 2.17 Saving a Superfile

---

Much data has been opened and changed, but nothing has been saved yet. When running a specific finite element model, the data needs to be saved in a specific format. However, SMS provides a generic format that can be used independent of the finite element model. Generic data includes the finite element mesh geometry, material settings, scatter point data, and the registered image. To save all this data for use in a later session:

1. Select *File / Save*. Each of the data types can be turned on or off. For this example, leave everything on.

2. In the *Prefix for all files* field enter the name *stmaryout* and click the *Update* button.
3. Click the *OK* button to save the files.

### 2.17.1 Saving Model Simulation Files

---

If you are using FESWMS or RMA2, you should save the simulation for use with later tutorials. To do this:

1. Select *FESWMS / Save Simulation* or *RMA2 / Save simulation*.
2. Enter the name *stmaryout* as the prefix and click the *Update* button.
3. Click the *OK* button to save the simulation.

## 2.18 Conclusion

---

This concludes the Overview of SMS tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## ***Mesh Editing***

This tutorial lesson teaches manual finite element mesh generation techniques that can be performed using *SMS*. It gives a brief introduction to tools in *SMS* which are useful for creating a finite element mesh from known topographical data points.

---

### **3.1 Importing Topographic Data**

Data points for a finite element mesh can be generated directly from topographic data, such as a list of survey points. An *XYZ* file contains the header *XYZ* on the first line of the file and then the *X*, *Y*, and *Z* coordinates of each point on a single line in the file. This type of file can be opened by *SMS*. To open the *poway1.xyz* file:

1. Select *File | Import*.
2. From the *Import Format* dialog, select the *XYZ Data* option and click the *OK* button.
3. Change to the *tutorial* directory and select the file *poway1.xyz*.

The data points from the file are converted to *mesh nodes*, points representing locations of known position and elevation. The data in *poway1.xyz* is shown in Figure 3-1. Any data represented by *X*, *Y*, and *Z* coordinates can be imported to *SMS* if it is in the *XYZ* file format. For file format details, see the *SMS Reference Manual*.

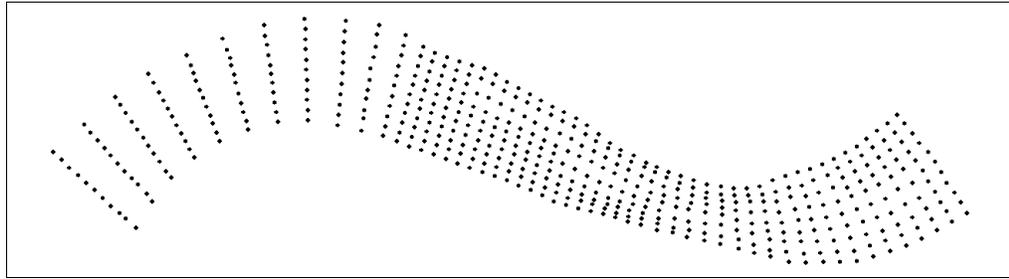


Figure 3-1 The poway1.xyz data points.

## 3.2 Triangulating the Nodes

---

After nodes have been created, elements are required to build a finite element mesh. Elements connect the nodes to define the extents of the flow area. *SMS* provides numerous automatic mesh generation techniques. This section will review a very simple technique, *triangulation*. To create a triangulated mesh from the data points:

1. Select *Elements / Triangulate*.
2. Since no nodes are selected, you will be prompted to triangulate all of them. Click the *Yes* button at this prompt.

When *SMS* triangulates data points, it creates either quadratic triangles or linear triangles from the mesh nodes. Different numerical models support different types of elements. *RMA2*, *FESWMS*, and *RMA10* support quadratic triangles, while *HIVEL*, *ADCIRC*, and *CGWAVE* support only linear elements. After the nodes are triangulated, the mesh will look like that in Figure 3-2. It may or may not have midside nodes, depending on whether the elements are linear or quadratic.

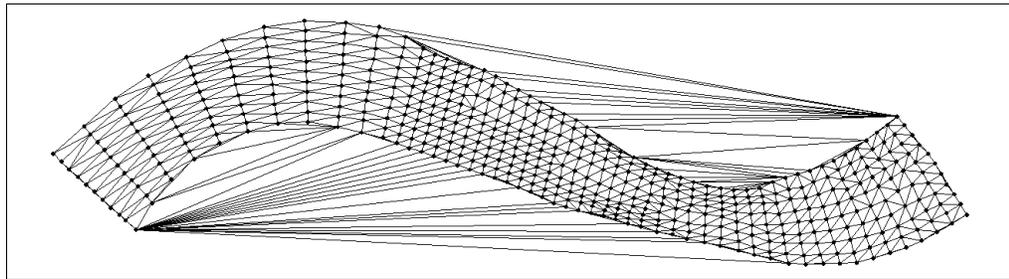


Figure 3-2. The results of triangulating the poway1.xyz data.

---

### 3.3 Deleting Outer Elements

---

The triangulation process always creates elements outside the desired mesh boundaries. For this tutorial, the mesh should be in the shape of a rotated *S*, so any elements outside of this boundary must be deleted. To remove these elements:

1. Choose the *Select Elements* tool  from the *Toolbox*.
2. Click on an element to select it.
3. Select another element by holding the *SHIFT* key and clicking on it.
4. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements.

It is tedious to individually select every element that needs to be deleted. *SMS* provides a hot key to help selecting groups of adjacent elements. To select a group of adjacent elements:

1. Choose the *Select Elements* tool  from the *Toolbox*.
2. Hold the *CTRL* key.
3. Click and drag a line through some elements to select them. Be careful to only select elements outside the *S* shape.
4. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements.

Continue deleting elements that are outside the boundaries of the *S* shape.

---

### 3.4 Deleting Thin Triangles

---

Many times, triangulation creates very thin triangular elements outside the desired mesh boundary. The three corner nodes of thin triangles are almost collinear and the elements may be too thin to see or select. If these are not deleted, large errors in the model solution can result.

*SMS* provides a way to define what is meant by a thin triangle using the element *aspect ratio*. The element aspect ratio is the ratio of the element width to its height. Perfect equilateral triangles have an aspect ratio of 1.0 while that of thin triangles is much less. To define the element aspect ratio:

1. Select *Elements | Options*.

2. Set the aspect ratio in the *Select Thin Triangle aspect ratio* box to 0.1. (The default value is 0.04). Triangular elements with an aspect ratio less than this are considered to be thin triangles.
3. Click the *OK* button.

The best aspect ratio to use for selecting thin triangles depends on the finite element mesh. For this mesh, the distribution of nodes is rather uniform, so a large aspect ratio will suffice. After this value is set, *SMS* can check for and select thin triangles. To delete any remaining thin triangles:

1. Select *Elements / Select Thin Triangles*. The lower right portion of the *Edit Window* shows how many elements became selected due to this operation, along with the total area of the selected elements. There may be quite a few elements selected.
2. Select *Edit / Delete* or press the *DELETE* key. If prompted to confirm, click the *Yes* button.

The mesh should now look like Figure 3-3.

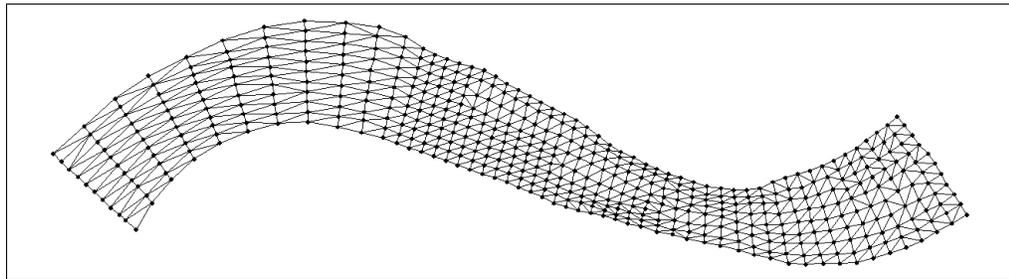


Figure 3-3. The *poway1* mesh after deleting excess triangles.

## 3.5 Merging Triangles

---

The mesh is composed entirely of triangles. Both *ADCIRC* and *CGWAVE* support only triangles. If you are using one of these models, you may skip this section of this tutorial.

Quadrilateral elements are generally preferred when using *RMA2*, *RMA10*, *FESWMS*, or *HIVEL* because:

- They make a more concise mesh for faster solutions.
- Quadrilateral elements are numerically more stable.

*SMS* can automatically merge a pair of triangles into a quadrilateral. Before merging triangles, the *Merge triangles feature angle* should be set. To do this:

1. Select *Elements / Options*.
2. Enter a value of 55.0 in the *Merge triangles feature angle* box (the default value is 65.0). Two triangles are allowed to be merged if all angles of the resulting quadrilateral are greater than the value specified.
3. Click the *OK* button.

The finite element method is more stable and accurate when quadrilateral elements are rectangular and triangular elements are equilateral. Although it is not practical for a mesh to exist entirely of these perfect shapes, the elements should approach these shapes as close as possible. For this reason, *SMS* merges triangles in an iterative manner. First, it merges elements using the angle criterion of  $90^\circ$ . Then, the angle criterion is decreased by a number of steps to the feature angle specified. By slowly decreasing the feature angle and testing all triangles against this, the best shaped elements will be formed from the merge command.

*SMS* can merge the triangles in either a selected portion of elements or all elements. In order to merge triangles in the entire mesh, no elements should be selected. To merge triangular elements into quadrilateral elements:

1. Select *Elements / Merge Triangles*.
2. Since no elements are selected, you will be prompted to merge all triangles. Click the *Yes* button at this prompt.

With most meshes, as is the case for this example, not all triangles will be merged. The mesh will appear as in Figure 3-4 after *SMS* merges the triangles.

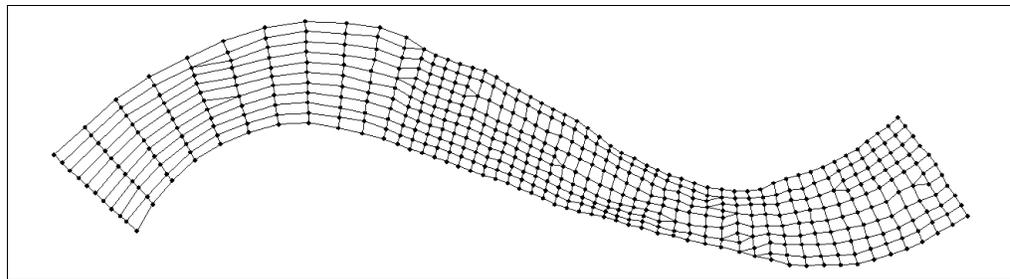


Figure 3-4. The poway1 mesh after merging triangles.

## 3.6 Editing Individual Elements

After triangulating the nodes, deleting elements outside the boundaries, and merging triangles, the mesh often needs further manipulation to add model stability. For a main river channel such as this model, lines of elements should run parallel to the mesh boundary. This is especially important in cases where a portion of the mesh may become dry so that the mesh will dry parallel to the boundary. Two of the tools

in SMS which are used for manipulating individual elements are the *Split / Merge*  tool and *Swap Edge*  tool. With the *Split / Merge* tool, two adjacent triangular elements can be merged into a quadrilateral element or a single quadrilateral element can be split into two triangular elements. With the *Swap Edge* tool, the common edge of two adjacent triangular elements can be swapped. See the *SMS Reference Manual* for a better description of these tools.

### 3.6.1 Using the Split / Merge Tool

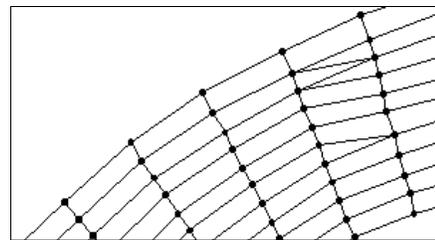
Most triangular elements in this mesh were merged into quadrilateral elements when the *Merge Triangles* command was performed in section 3.5. Some of the elements that were not automatically merged can be merged manually. To do this:

1. Zoom into the portion of the mesh shown in Figure 3-5a. Notice the two triangular elements separated by a number of quadrilateral elements.
2. Select the *Split / Merge* tool  from the *Toolbox*.
3. Split the quadrilateral, highlighted in Figure 3-5a, by clicking inside it. There should now be three triangles, as shown in Figure 3-5b.
4. Merge the top two triangles, highlighted in Figure 3-5c, by clicking on the edge between them. (The *Split / Merge* tool should still be selected.)

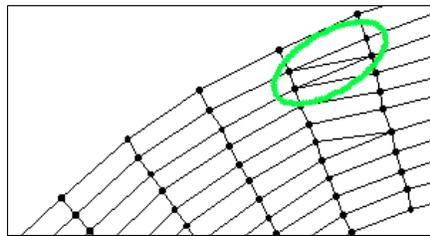
The result of this split/merge operation is shown in Figure 3-5d. There is now one fewer quadrilateral between the two lonely triangles.



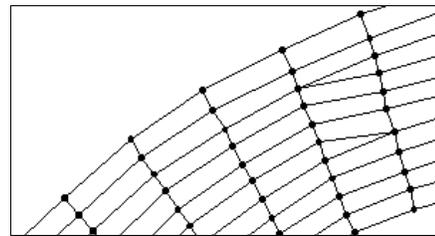
(a). Initial elements.



(b). After splitting quadrilateral.



(c). After swapping edge.



(d). Final elements.

Figure 3-5. Example of manual split / merge procedure.

To finish editing this section:

- Repeat the above split/merge process until there are no more triangles across the section. This part of the mesh should look like Figure 3-6.

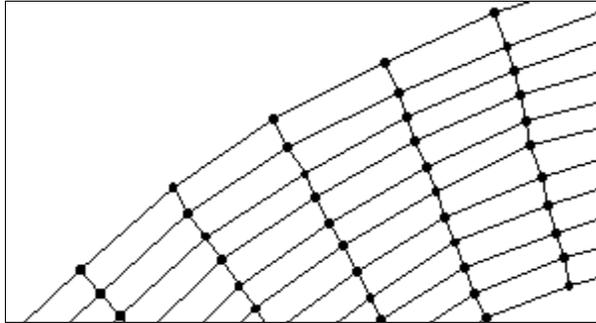


Figure 3-6. The mesh section after merging triangles.

### 3.6.2 Using the Swap Edge tool

The common edge between two triangles can be swapped. The best way to understand this is to think of the two triangles as a quadrilateral, and the common edge between them is a diagonal of the quadrilateral. By swapping this common edge, it changes to be along the opposite diagonal of the quadrilateral. If this edge is clicked again, it returns back to its original state. This can be seen in Figure 3-7.

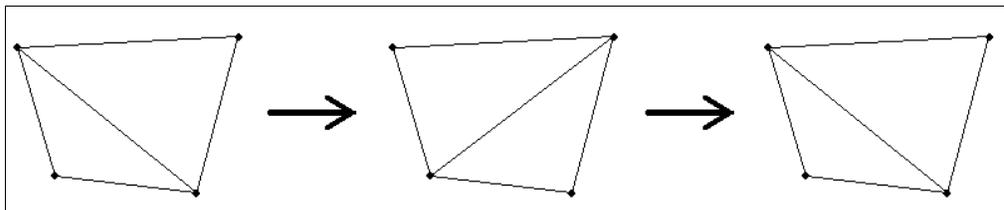


Figure 3-7 The Swap Edge technique.

One place in this mesh requires the use of the *Swap Edge* tool together with the *Split / Merge* tool to be able to merge the triangles. This is located toward the middle of the finite element mesh, at the constriction. The easiest way to find this location is to set the window boundaries to the correct location. To do this:

1. Select *Display / Set Window Boundaries*.
2. In the *Set Window Boundaries* dialog, select to use the *X range to be specified* option.
3. Enter these values: *X at left* = 25,200; *X at right* = 25,500; *Y at bottom* = 9300.
4. Press the *OK* button.

You should now be able to see the portion of the finite element mesh shown in Figure 3-8. In this part of the mesh, there are two triangles that need to be merged together, separated by a single quadrilateral. To do this:

1. Choose the *Split / Merge*  tool from the *Toolbox*.
2. Click inside the quadrilateral, highlighted in Figure 3-8a, that separates the two triangles. The quadrilateral gets split as shown in Figure 3-8b. The new edge was not created in the direction necessary to merge the outer triangles.
3. Choose the *Swap Edge*  tool from the *Toolbox*.
4. Click only once, directly on the edge that was just created inside the quadrilateral. The edge will swap to the other diagonal of the quadrilateral. This result is shown in Figure 3-8c.
5. Once again choose the *Split / Merge* tool from the *Toolbox*.
6. Merge the top two triangles to form one quadrilateral, and then merge the bottom two triangles to form another quadrilateral. The result is shown in Figure 3-8d.

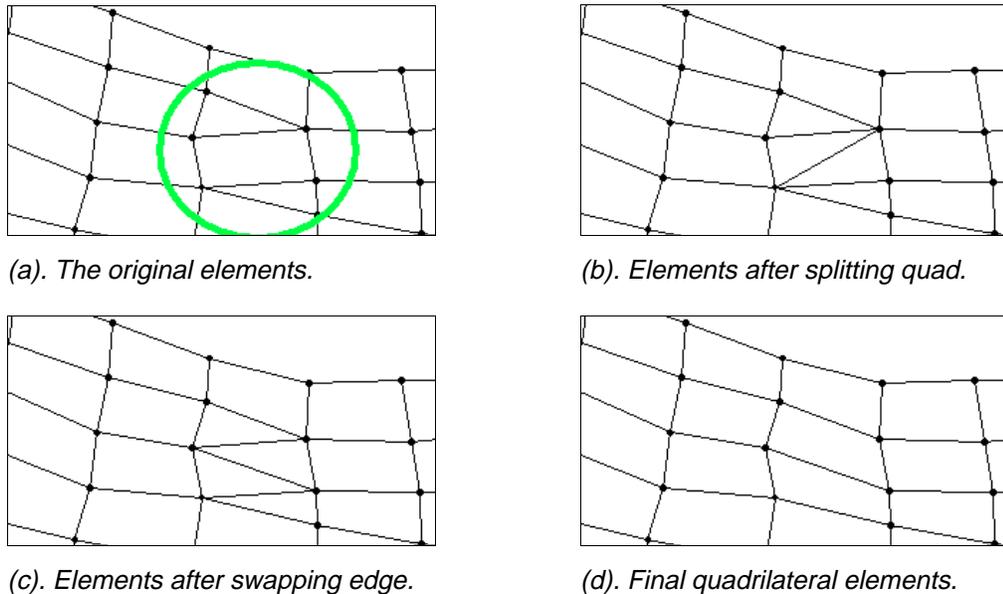


Figure 3-8 Example of manual swapping procedure.

Although this operation appears simple, it is one that takes some time to get used to performing. Most people do not get through this without making a mistake. However, after you understand this operation, it is easier to use. The *Split / Merge* and *Swap Edge* tools are very useful for manually adjusting small areas of the finite element mesh.

Continue to merge triangles in the areas that you are able to do so. Not all of the triangles can be merged. When you are done, there should be only six triangles left in the finite element mesh, and it should look like that shown in Figure 3-9.

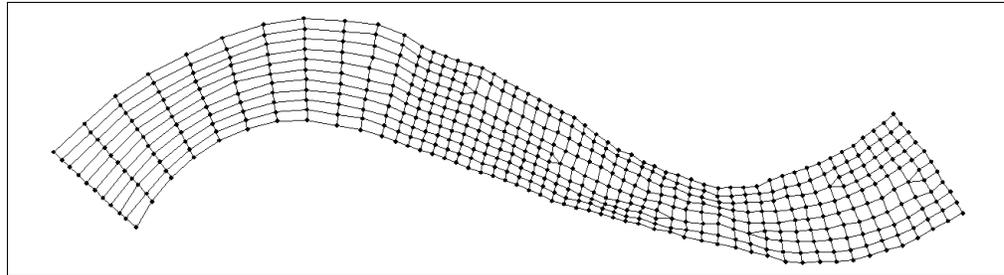


Figure 3-9 The finite element mesh after merging triangles.

### 3.7 Smoothing the Boundary

When dealing with quadratic finite element meshes, mass loss can occur through a jagged boundary. It is good to smooth the boundary of a quadratic mesh to prevent these losses. Smoothing can only be performed with quadratic models, because the midside nodes get moved while corner nodes do not. *SMS* currently supports three quadratic finite element models, *RMA2*, *RMA10*, and *FLO2DH*. If you are not using one of these quadratic models, you can skip this section. The quadratic models still support the creation of linear elements. To make sure you have quadratic elements:

1. Select *File / Get Info*.
2. In the top right corner of the *Mesh Information* dialog, look at the *Element type* defined. This will be either *quadratic* or *linear*.
3. Click the *Close* button in the *Mesh Information* dialog.
4. If the element type was *linear*, select *Elements / Linear <-> Quadratic* to switch the element type. If it was *quadratic*, you are already set.

The easiest way to smooth the entire mesh boundary is by creating a nodestring around the entire mesh boundary. To do this:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click the node labeled *Node 1* in Figure 3-10.
3. Hold the *CTRL* key and double-click the node labeled *Node 2* in Figure 3-10. When holding the *CTRL* key, *SMS* creates a nodestring counter-clockwise around the mesh boundary from the first node to the second node.

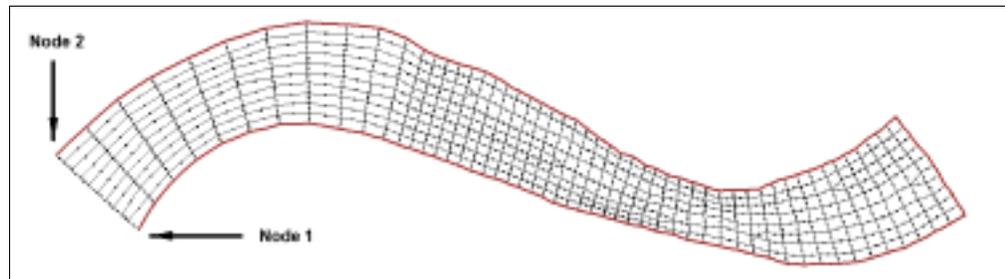


Figure 3-10 The nodestring to create for smoothing.

This nodestring starts from *Node 1*, and runs counter clockwise around the entire boundary to *Node 2*. Notice that this nodestring goes around two sharp corners on the right side of the mesh. To assure that these corners remain sharp:

1. Select *Elements / Options*.
2. In the *Element Options* dialog, change the *Smooth nodestring feature angle* to be 45.0. A midside node will not move if it is at a corner that is sharper than this angle.
3. Click the *OK* button.

Now that the nodestring is created and the feature angle is set, the boundary is ready to be smoothed. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. A small icon will appear at the center of the nodestring.
2. Click on the icon to select the nodestring. The icon will be filled and the nodestring will be highlighted in red.
3. Select *Elements / Smooth Nodestring*. The mesh boundary will be smoothed as shown in Figure 3-11.

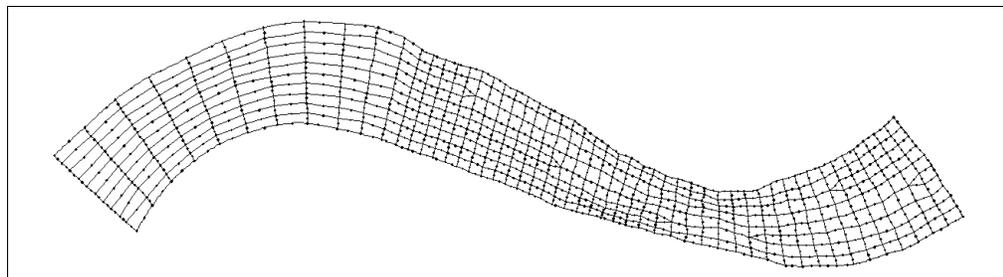


Figure 3-11. Example nodestrings for smoothing the mesh boundary.

In general, it is sufficient to smooth the finite element mesh boundary. However, it may be desirable to further smooth interior elements at sharp bends or where dry elements may change the boundary. Any nodestring can be used for smoothing. See

the *SMS Reference Manual* for more information on creating interior nodestrings and the smoothing operation.

### 3.8 Renumbering the Mesh

The process of creating and editing a finite element mesh, as performed in the previous few sections, causes the node and element ordering to become disorganized. You can see this disorganization by refreshing the display. To do this:

- Select *Display / Refresh* or click the *Refresh*  macro in the *Toolbox*.

Notice that the elements are drawn on the screen in a random order. This random mesh ordering increases the size of the matrices required by the finite element analysis codes. A good mesh ordering can be restored by renumbering the mesh. Renumbering starts from a nodestring. To renumber this mesh:

1. Choose the *Create Nodestring* tool  from the *Toolbox*.
2. Create a nodestring across the left section, as shown in Figure 3-12.
3. Choose the *Select Nodestring* tool  from the *Toolbox* and select the nodestring that was just created.
4. Choose *Elements / Renumber*. You can choose either the *Band Width* or *Front Width* option for this mesh. For more information on these two options, see the *SMS Reference Manual*.
5. Click the *OK* button to start the renumbering process.

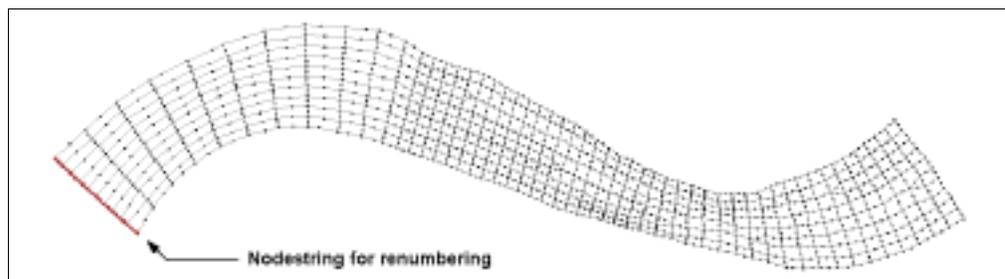


Figure 3-12. The position of the nodestring for renumbering.

When *SMS* is finished renumbering the mesh, the display will refresh. As it does, the new node and element ordering can be seen. Notice that the elements and nodes are in an ordered fashion starting from the location of the nodestring.

Remember that adding and deleting nodes or elements changes the mesh order. It is important that renumbering be the last step of the mesh creation process. Editing a

mesh invalidates any boundary condition and/or solution files that have previously been saved. (Boundary condition and solution files are discussed in later tutorials).

### 3.9 Changing the Contour Options

---

When the finite element mesh is created, contours lines are drawn to connect points of equal elevation. By default, these contours are displayed as constant green lines. The contour display can be changed using the *Contour Options* dialog. It is always a good idea to look at a color contour map after a new finite element mesh has been created. This helps you better visualize the bathymetry of the model. To set the color fill contours:

1. Choose *Data / Contour Options* or click the *Contour Options*  macro.
2. In the *Contour Method* section of the *Contour Options* dialog, select the *Color fill between contours* option.
3. Click the *OK* button.

The display will refresh with color filled contours such as those shown in Figure 3-13. If you do not see color filled contours, then the display of contours has been turned off. To turn the contour display back on:

1. Choose *Display / Display Options* or click the *Display Options*  macro.
2. In the *Mesh Display Options* dialog, turn on the *Contours* option.
3. Click the *OK* button.

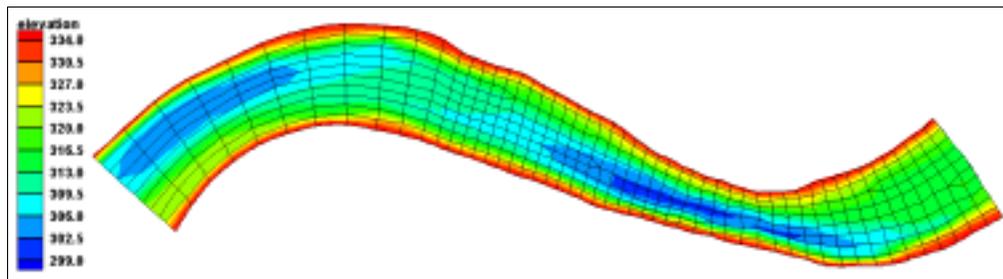


Figure 3-13 Elevation contours of the poway1 mesh.

In this plot, you can see that there are two pits in the river, while both banks are the highest part. For more examples of how to work with display and contour options in SMS, see the *SMS Reference Manual*.

### 3.10 Checking the Mesh Quality

Another important thing to check with a newly created finite element mesh is the mesh quality. There are various things that *SMS* looks at when checking this. To turn on the mesh quality:

1. Select *Display / Display Options* or click the *Display Options*  macro.
2. Turn off the *Contours* option.
3. Turn on the *Mesh quality* option.
4. Click the *OK* button.

The display will refresh without contours and with the mesh quality, as shown in Figure 3-14. The mesh quality shows where problem areas may occur. A legend shows which color corresponds with which quality item. See the *SMS Reference Manual* for more information on these mesh quality options.

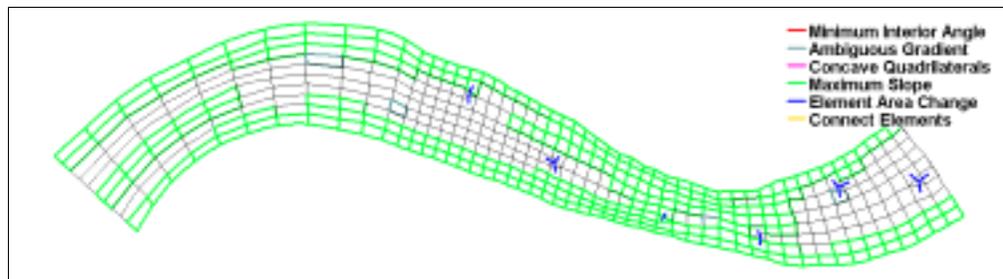


Figure 3-14 Mesh quality for the Poway1 finite element mesh.

Many elements are highlighted because of the maximum slope warning. Elements that are steep in the flow direction may cause supercritical flow to occur. In this mesh, however, the elements are steep in the direction perpendicular to the flow, so this warning can be ignored. To turn off this mesh quality check:

1. Select *Display / Display Options* or click the *Display Options*  macro from the *Toolbox*.
2. In the *Mesh Display Options* dialog, click the *Options* button next to the *Mesh quality* item.
3. In the *Element Quality Checks* dialog, turn off the *Maximum Slope* option.
4. Click the *OK* button in both dialogs.

Once again, the display will refresh (see Figure 3-15), but this time, no slope warnings will be shown. There are only two warning types that remain. The *Element Area Change* warning is not crucial, especially when there are not many of these

warnings in the mesh. For this mesh, this element quality warning will be ignored. The *Ambiguous Gradient* warning is shown for four elements, which are numbered in the figure. If the flow through these elements is deep, then the ambiguous gradient will not really effect the flow pattern. However, if the flow through these elements is shallow, the element can become an artificial dam. Since the flow depths are not yet known for this finite element mesh, these quality problems will be fixed.

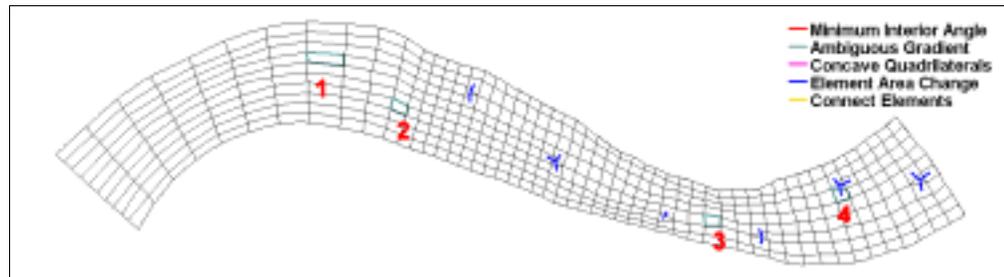


Figure 3-15 Mesh Quality without the Maximum Slope quality check.

If the ambiguous gradient is not very large, it can be fixed by slightly modifying nodal elevations. This is the case with elements 1, 2, and 4 in the figure. A large ambiguous gradient, which would require editing nodal elevations by more than a foot or two, should just be split into two triangular elements. This is the case with element 3 in the figure.

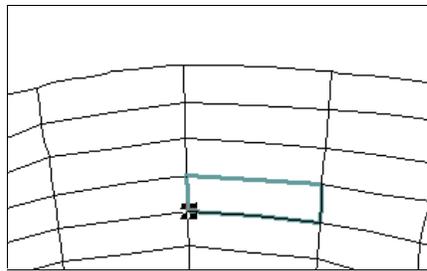
Before changing the nodal elevations, you should set the *Edit Window* z-step value. In the *Edit Window*, there are two arrows next to the *Z Coordinate* box. When a node is selected, these arrows increase and decrease the selected node elevation value. By default, the increment is automatically computed. However, you often want to control this. To set the increment value:

1. Select *Nodes / Options*.
2. In *Nodal Z-Value Options* section of the *Node Options* dialog, select the *User-defined Z step* option.
3. Set the step value to 0.1. This assures that elevation values are only changed by 0.1 feet with the up and down arrows in the *Edit Window*.
4. Click the *OK* button to close the *Node Options* dialog.

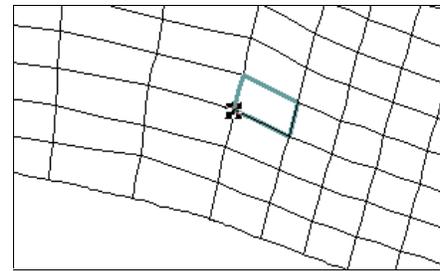
With the *Z* step set to a small value, the ambiguous gradients can be removed. To do this:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. For element 1, select the bottom left corner node, as shown in Figure 3-16a. Increase its elevation by 0.5 feet by clicking the up arrow  in the *Edit Window* five times. (Remember that the increment was set to 0.1 feet.)

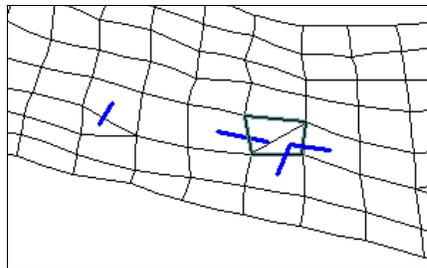
3. For element 2, select the bottom left corner node, as shown in Figure 3-16b. Increase its elevation by 0.2 feet.
4. For element 3, split and swap the quadrilateral as necessary so that it looks like Figure 3-16c.
5. For element 4, select the upper right corner node, as shown in Figure 3-16d. Increase its elevation by 0.2 feet.



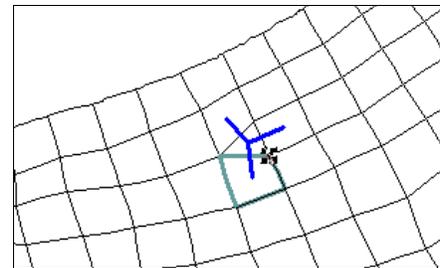
(a). Select node on element 1.



(b). Select node on element 2.



(c). Split element 3.



(d). Select node on element 4.

Figure 3-16 The four quadrilateral elements with an ambiguous gradient.

After making these modifications, you will not have any more . ambiguous gradient warnings in the finite element mesh. The following three things should now be done (in no particular order):

- *Turn off the display of element quality checks.* You are done looking at the mesh quality, so this should be turned off to make the screen less busy.
- *Turn on the display of color filled contours.* Check that the adjustments you made did not make funny looking contours in the mesh. When editing nodal elevation values, it is always important to check the contours. If funny looking contours result, you may want to put things back the way they were and make some different changes.
- *Renumber the mesh.* Remember, whenever you adjust the finite element mesh, it should be renumbered. If you had only modified elevation values, then the mesh would not require renumbering. However, you must renumber the mesh after splitting a quadrilateral into two triangles.

### 3.11 Saving the Mesh

---

If *SMS* is registered, then the finite element mesh can be saved. This mesh will not be used in other tutorials, so saving it is not required. To save the mesh:

1. Select *File / Save*.
2. Turn everything off except for the *2D Mesh* option.
3. Enter a *Prefix for all files* name of *poway1* and click the *Update* button.
4. Click the *OK* button.

### 3.12 Creating Rectangular Patches

---

In the previous sections, you worked on creating a finite element mesh from regularly spaced survey data points. Although this created a very nice mesh, you do not usually have such data. Many times, your survey data consists of just cross sections and the bank lines. When this is the case, you can use another mesh generation technique, *Rectangular Patches*.

A rectangular patch is defined using four nodestrings. These nodestrings should form a continuous loop, and be shaped much like a rectangle. Rectangular patches are very nice for creating elements in the main flow channel of a river because the elements of the patch will automatically be aligned with the flow direction. This decreases the need for manual splitting and swapping of elements, as was done in section 3.6.

#### 3.12.1 Gathering Cross Section Data

---

When gathering cross section data in the river, it is imperative to locate important river features. This will be illustrated in this section. First, a new xyz file should be opened. To do this:

1. Select *File / Import*.
2. From the *Import Format* dialog choose the *XYZ Data* option and click *OK*.
3. Find and select the file *poway2.xyz* from the *tutorial* directory. You will be prompted to replace or append the existing data. Click the *Replace* button.

This data file contains far less points than the one opened in section 3.1. This data is shown in Figure 3-17.

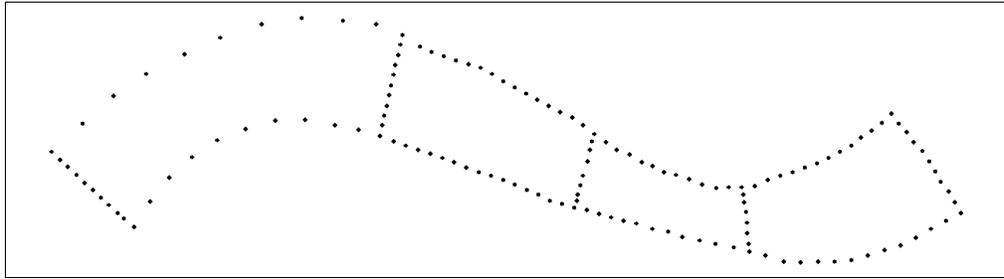


Figure 3-17 The poway2.xyz data.

One of the benefits of using rectangular patches is that they automatically create interior nodes and elements. This means that you can use less survey points, decreasing the effort required to obtain them.

For a rectangular patch, you must have four nodestrings forming a continuous loop. To create the nodestrings for the first patch:

1. Choose the *Create Nodestring* tool  from the *Toolbox*.
2. Create the first nodestring by clicking from node to node as shown in Figure 3-18a. Remember to terminate the nodestring by double clicking the last node. *NOTE: You may not use the SHIFT key or CTRL key because the nodes are not connected by elements. You must click on every node!*
3. Create the second nodestring as shown in Figure 3-18b.
4. Create two more nodestrings, to finish the nodestring loop as shown in Figure 3-18c.

After the nodestrings have been created, they may be used to create a rectangular patch. To create the patch:

1. Choose the *Select Nodestrings* tool  from the *Toolbox*. Notice that a small icon appears at the center of each nodestring.
2. Select all four nodestrings by dragging a box around all these icons. Each nodestring gets highlighted in red.
3. Select *Elements / Rectangular Patch*. The *Rectangular Patch Options* dialog shows the layout of each of the selected nodestrings.
4. For this patch, use the default options. Click the *OK* button.

When the *Rectangular Patch Options* dialog is closed, a patch is created in the area specified by the selected nodestrings. If, for some reason, linear elements were created, turn the elements to quadratic by selecting *Elements / Linear<->Quadratic*. The geometry should now appear as Figure 3-18d.

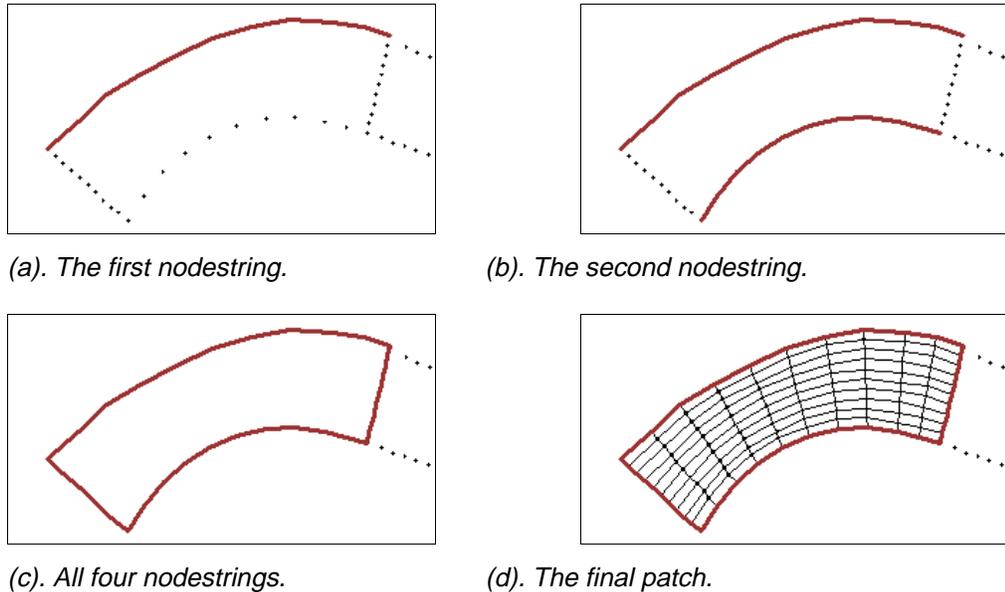


Figure 3-18. Step-by-step creation of a patch.

This patch looks like it is okay. The elements that were created are aligned with the flow direction and they are all quadrilaterals. However, you will see a problem if you turn on the color filled contours. To turn on the contours:

1. Choose *Display / Display Options* or click the *Display Options*  macro from the *Toolbox*.
2. In the *Mesh Display Options* dialog, turn on the *Contours* item.
3. Click the *Options* button next to the *Contours* item.
4. In the *Contour Method* section of the *Contour Options* dialog, select the *Color fill between contours* option.
5. Click *OK* in both dialogs.

The display will refresh with color filled contours such as those shown in Figure 3-19.

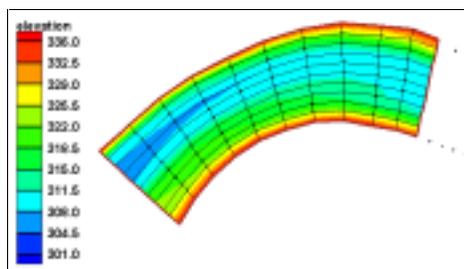


Figure 3-19 Contours on the patch.

While these contours look okay, there is something missing. If you compare these contours with those in Figure 3-13 on page 3-12, you will see that a large scoured area exists in the outer part of the bend. Remember that the interior nodes of the patch are interpolated from the boundary data. Since there was no cross section through this scoured area, it does not show up in the elements of the patch.

This is why you must be very careful when gathering cross sectional data. Although it saves effort by gathering less data points, you can lose important features that should be included in your finite element mesh. If your geometry data is not accurate, then you will not get accurate results from the finite element model.

### 3.12.2 Creating the Mesh with Rectangular Patches

As was shown in section 3.12.1, you must be careful when you gather cross sectional data for use with rectangular patches. This river data will be better if two more cross sections are shot, one through the middle of each of the scoured areas. An updated data file that has these extra cross sections can be opened. To do this:

1. Select *File / Import*.
2. From the *Import Format* dialog choose the *XYZ Data* option and click *OK*.
3. Find and select the file *poway3.xyz* from the *tutorial* directory. You will be prompted to replace or append the existing data. Click the *Replace* button.

This data (see Figure 3-20) contains two cross sections more than what was in the previous data. Although more survey points had to be gathered, the resulting finite element mesh should be more representative of the river.

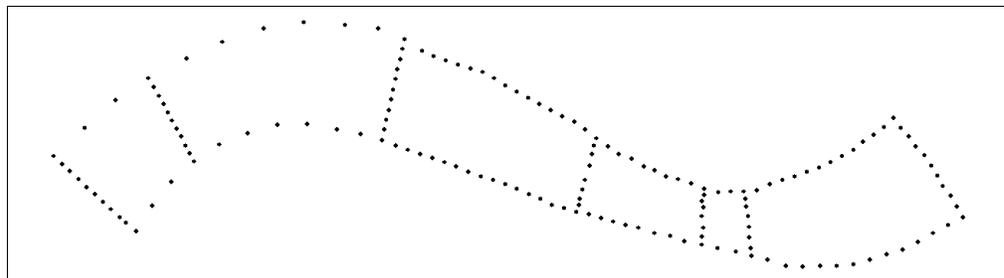


Figure 3-20 The *poway3.xyz* cross section and bank data.

You are already familiar with the patch creation process, so this will not be explained in great detail. This data requires the creation of six rectangular patches. Nodestrings must be created across each cross section. In addition, six nodestrings must be created along each bank. Remember that the four nodestrings for a rectangular patch must form a continuous loop. To show this:

- Create the three patches shown in Figure 3-21. In each case, use the default values in the Rectangular Patch Options dialog.

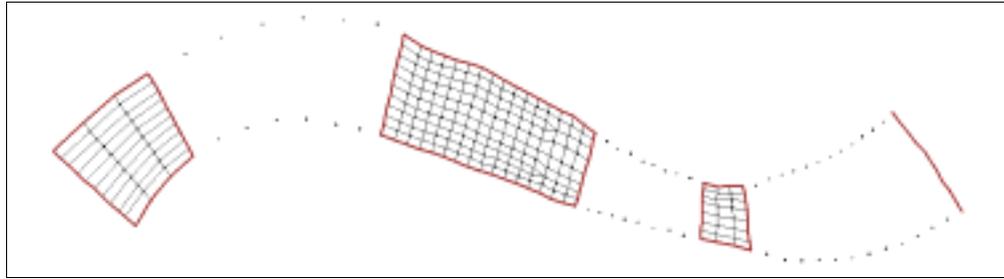


Figure 3-21 The first three patches from the poway3 data.

Refer back to the previous section if you need assistance in creating these patches.

After these have been created, create the next three patches. To do this:

1. Create the rest of the nodestrings along the banks. You do not need to duplicate cross section nodestrings that were created for the first patches.
2. Using these nodestrings, create the other three patches. The final mesh is shown in Figure 3-22 (the thick lines are the nodestrings).

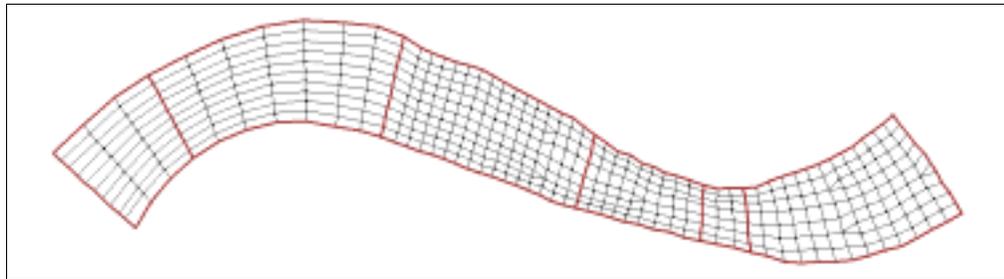


Figure 3-22 The mesh with all patches created.

This finite element mesh is very similar to the one created using triangulation. This mesh, however, required less manual manipulation and less survey data than the first one. To see how the contours for this finite element mesh compare with those of the triangulated mesh:

1. Choose *Display / Display Options* or click the *Display Options*  macro from the *Toolbox*.
2. In the *Mesh Display Options* dialog, turn on the *Contours* item.
3. Click the *Options* button next to the *Contours* item.
4. In the *Contour Method* section of the *Contour Options* dialog, select the *Color fill between contours* option.

5. Turn on the *Contour between specified range* option. With a *Minimum value* of 299 and a *Maximum value* of 334. Setting the range to the same one used by the first contour map assures that this contour map is comparable.
6. Click *OK* in both dialogs. The resulting contours are shown in Figure 3-23.

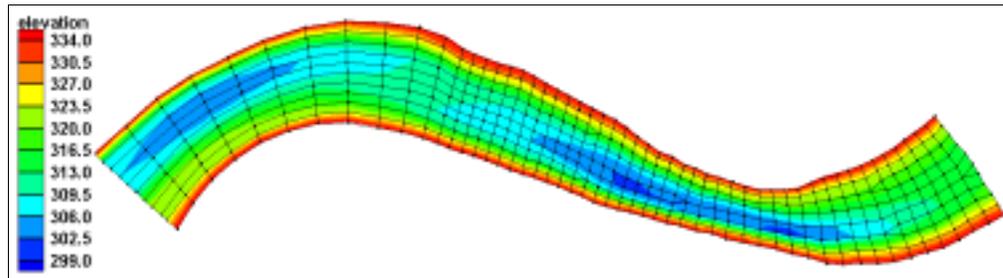


Figure 3-23 Contours of the poway3 mesh.

Comparing this color plot with that of Figure 3-13 on page 3-12, only slight variations can be seen. You would expect a better definition of the river with more survey data points. However, the variations are so slight that this second finite element mesh is good enough, especially since it required less effort to create.

### 3.12.3 A Note on Rectangular Patches

When creating rectangular patches, the number of nodes on opposite sides of the patch do not need to be the same. However, the best patches will have the same number of nodes on both river banks. This was the case for all the patches created in section 3.12.2.

Another thing that happens when creating many rectangular patches is that there are many nodestrings created. Most of these are unnecessary for further mesh modifications. To delete all the nodestrings:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. You will see a small icon appear at the center of each nodestring.
2. Select *Edit / Select All*. This will select all the nodestrings.
3. Click the *Delete*  macro or select *Edit / Delete*.

## 3.13 Refining Elements

At times, it is desirable to refine part of a mesh so that there is more definition in that area. More definition helps to increase accuracy and decrease divergence problems. It is important to not refine too much of a mesh, however, because more nodes and

elements increase the time required for finite element computations. In this section, you will refine the section of elements on the left edge of the mesh.

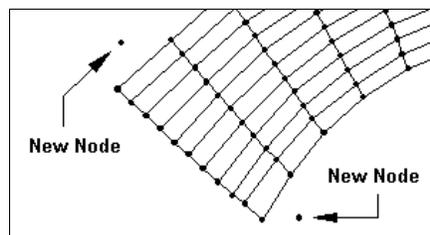
### 3.13.1 Inserting Breaklines

The elements at the left of the mesh are already rather skinny. The first refinement will be to cut them across their width. This can be done using a nodestring as a breakline. As you have seen previously, a nodestring must be created using existing nodes. Therefore, before being able to create a nodestring to cut the elements, you must create two nodes, one on either side of the channel. To do this:

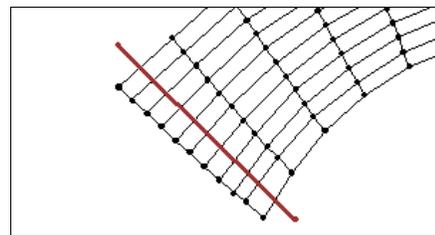
1. Choose the *Create Nodes*  tool from the *Toolbox*.
2. Click once on each side of the channel, near the middle of the left-most column of elements, as shown in Figure 3-24a.

A nodestring can now be created from one of these new nodes to the other. This nodestring will be used as a breakline. To create the nodestring:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click on one of the new nodes. Double-click on the other. The nodestring will appear, as shown in Figure 3-24b.



(a). Two nodes to create.



(b). The nodestring to create.

Figure 3-24 Adding the nodestring to use as a breakline.

With the nodestring created, it can be used as a breakline. A breakline splits all the elements that it crosses, forcing element edges to appear along the line. To make a breakline from the nodestring:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the icon that appears in the center of the nodestring.
3. Select *Elements | Add Breaklines*. The elements will be split along the nodestring, as shown in Figure 3-25.

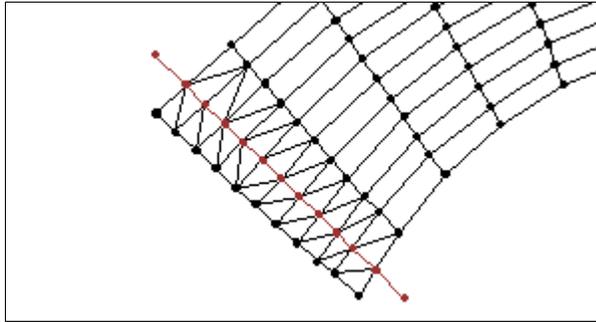


Figure 3-25 The breakline has been inserted.

Now that the nodestring has been used as a breakline, it is no longer needed. It should still be selected. To remove the nodestring:

- Select *Edit / Delete* or click the *Delete*  macro.

When the elements get broken along the breakline, triangular elements are created. These should be merged into quadrilateral elements. To do this:

1. Choose the *Select Elements*  tool from the *Toolbox*.
2. Select *Edit / Select With Poly*. This allows you to select a specific set of elements by drawing a polygon around them.
3. Click out a polygon that surrounds all the triangular elements that were created by the breakline. Double-click to end the polygon.
4. With the triangular elements highlighted, select *Elements / Merge Triangles*.

All of the triangular elements that were created by the breakline will be merged into quadrilateral elements. With these elements created, you just need to get rid of the two nodes that were created to define the breakline. These nodes are not connected to any elements, and are thus called *disjoint*. To remove the disjoint nodes:

1. Select *Nodes / Select Disjoint Nodes*. You should get a message that two disjoint nodes were found and selected. Click *OK* to this prompt.
2. Select *Edit / Delete* or click the *Delete*  macro.

Now that the breakline has been inserted, triangular elements have been merged into quadrilaterals, and the disjoint nodes have been deleted, the mesh should look like that in Figure 3-26.

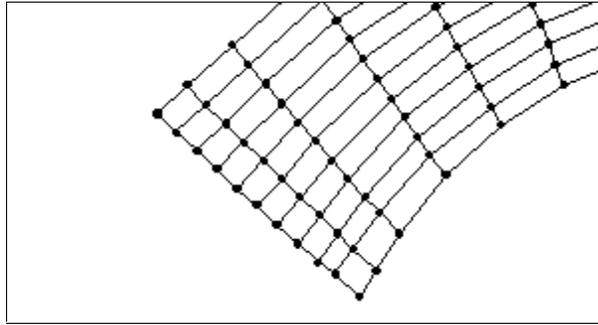
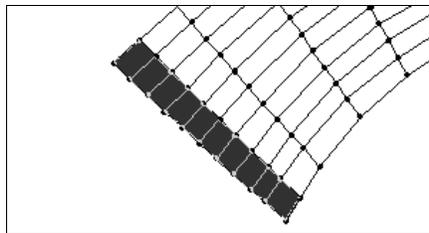


Figure 3-26 The final mesh after inserting the breakline.

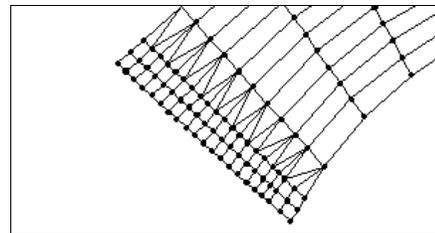
### 3.13.2 Using the Refine Command

Now that the breakline has been inserted, you are ready to use the refine command. This command splits a quadrilateral element into fourths. The reason the breakline was added is so that the refined elements would not be too skinny. You will refine the first column of elements on the very left side. To refine these elements:

1. Choose the *Select Elements*  tool from the *Toolbox*.
2. Hold the *CTRL* key and drag a line through the left-most column of elements, as shown in Figure 3-27a.
3. Select *Elements / Refine*. Each of the selected quadrilaterals will be split into four smaller quadrilaterals, and triangles will transition these small quadrilaterals to the larger quadrilaterals, as shown in Figure 3-27b.



(a). The elements to select.



(b). After the refine command.

Figure 3-27 The section of the mesh to refine.

## 3.14 Finishing the Mesh

Now that elements have been created and edited, the following things should be done before using this mesh in a finite element analysis:

- The Mesh Quality should be checked. You will see the same types of warnings as in section 3.10.

- The mesh should be renumbered. Remember that whenever nodes and elements are created, the mesh order should be fixed as in section 3.8.

If you wish, you may now save this data (see section 3.11). However, this is not required since this mesh will not be used in further tutorials.

### **3.15 Conclusion**

---

This concludes the Mesh Editing tutorial. Although not every option was discussed, you should be familiar with many of the tools that *SMS* provides for mesh generation and editing. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## ***Basic RMA2 Analysis***

---

### **4.1 Introduction**

---

This lesson will teach you how to prepare and run an *RMA2* analysis. You will be using the file *stmaryout.sim* which was created in lesson 2. The geometry should have been created and renumbered in the earlier lesson. If you did not do this session, you can open the simulation that is supplied in the *tutorial/output* directory. To open the simulation file:

1. Select *RMA2 | Open Simulation*.
2. Open the file *stmaryout.sim*, either the one that you created in lesson 2 or the one from the *tutorial/output* directory.
3. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The flow and head boundary conditions were automatically applied by the conceptual model when the mesh was generated.

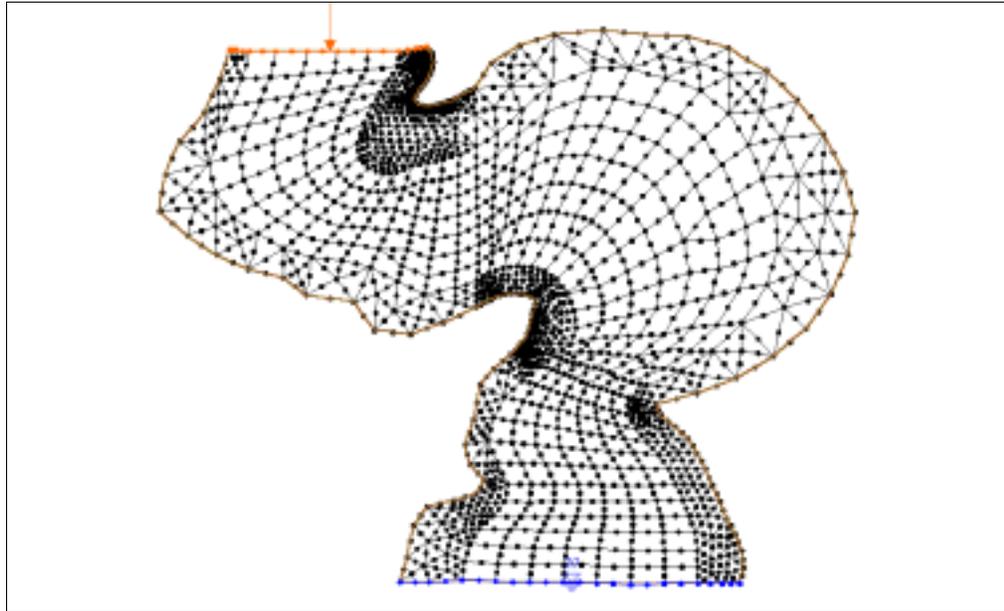


Figure 4-1. The mesh from *stmaryout.sim*.

## 4.2 Defining Material Properties

---

Each element of the mesh is assigned a material type ID. When the file was opened, the materials were created with default parameters. These material properties must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Reference Manual* for details of what each parameter represents). To edit the material parameters:

1. Select *RMA2 / Material Properties*.
2. In the *RMA2 Material Properties* dialog, highlight the material *Material 1*.
3. Make sure the *Isotropic eddy viscosities* option is turned on and enter a value of 25 for the eddy viscosity ( $E$ ).
4. Set the Manning's roughness value ( $n$ ) to 0.020.
5. Highlight the material *Material 2* set  $E$  to 25 and  $n$  to 0.03.
6. For the material labeled *Material 3*, assign an  $E$  of 30 and an  $n$  of 0.003.
7. Click the *Close* button to close the *RMA2 Material Properties* dialog.

The material properties have now been properly defined. Note: the material zones can be displayed by opening the *Display Options* dialog and turning on the *Materials* option.

### 4.3 Checking The Model

---

Before running an analysis, the model should be checked for completeness. SMS provides a model checker for each numerical model that it supports. Although passing the model checker will not guarantee that the model will converge, some of the more common mistakes will be reported. To run the model checker:

1. Select *RMA2 / Model Check*.
2. Click the *Run Check* button.

The *RMA2* model checker will report a warning that the mesh has not been renumbered during this session. In this case, the warning can be ignored because the model was already renumbered in a previous session.

### 4.4 Saving The Simulation

---

The flow and head boundary conditions were previously defined inside the map module. The entire simulation can now be saved. To save the simulation:

1. Select *RMA2 / Save Simulation*.
2. Enter the name *stmary2* for the *Simulation file* and click the *Update* button.
3. Click the *OK* button to save the simulation.

After saving the simulation, the analysis can be run.

### 4.5 Using GFGEN

---

Before running the finite element analysis, the ASCII geometry file created by *SMS* must be converted to a binary format that *RMA2* can understand. This is done with a program called *GFGEN*. To launch the *GFGEN* program:

1. Select *RMA2 / Run GFGEN*.
2. If a message such as “gfgv435.exe – not found” is given, then click the *File Browser* button  to manually find the *GFGEN* executable.
3. Click the *OK* button to launch *GFGEN*.

When the *GFGEN* window finishes, a beep will sound and you will get a message to press the *RETURN* key. If you do not get such a message and the window goes away, then *GFGEN* crashed and you may want to contact technical support with your file.

## 4.6 Using RMA2

---

After *GFGEN* is successfully completed, the finite element analysis can run. *RMA2* is the analysis program which computes 2D flow solutions at each node. For the mesh used in this tutorial, a steady state solution will be computed. To launch the *RMA2* program:

1. Select *RMA2 / Run RMA2*. As with *GFGEN*, the prompt shows the location of the most recent *RMA2* executable.
2. If necessary, find the executable, and click the *OK* button to launch *RMA2*.

For this steady state simulation, *RMA2* should finish in a couple of minutes. When the simulation is finished, a beep will sound and you will be prompted to press the *RETURN* key. The file *stmary2.sol* will contain the *RMA2* solution data.

## 4.7 Conclusion

---

This concludes the Basic *RMA2* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

Select *File / Quit*. If prompted to confirm, click the *Yes* button.

---

## ***Basic FESWMS Analysis***

---

### **5.1 Introduction**

---

This lesson will teach you how to prepare a mesh for a *FESWMS* simulation. You will be using the file *stmaryout.fil* which was created in lesson 2. If you have not completed this lesson, the file needed for this tutorial can be found in the *output* directory. A *FESWMS .fil* file is a type of super file. It contains a list of filenames that are used by *FESWMS*. The actual input data is stored in the files named in the super file. To open the file:

1. Select *Data / Switch Current Model*.
2. From the *Current Model* Dialog, choose *FESWMS* and click the *OK* button.
3. Select *FESWMS / Open Simulation*.
4. Open the file *stmaryout.fil*, either the one saved from lesson 2 or the one supplied in the *tutorial/output* directory. The display will refresh with the mesh as shown in Figure 5-1.

The flow and head boundary conditions were automatically applied by the conceptual model when the mesh was generated.

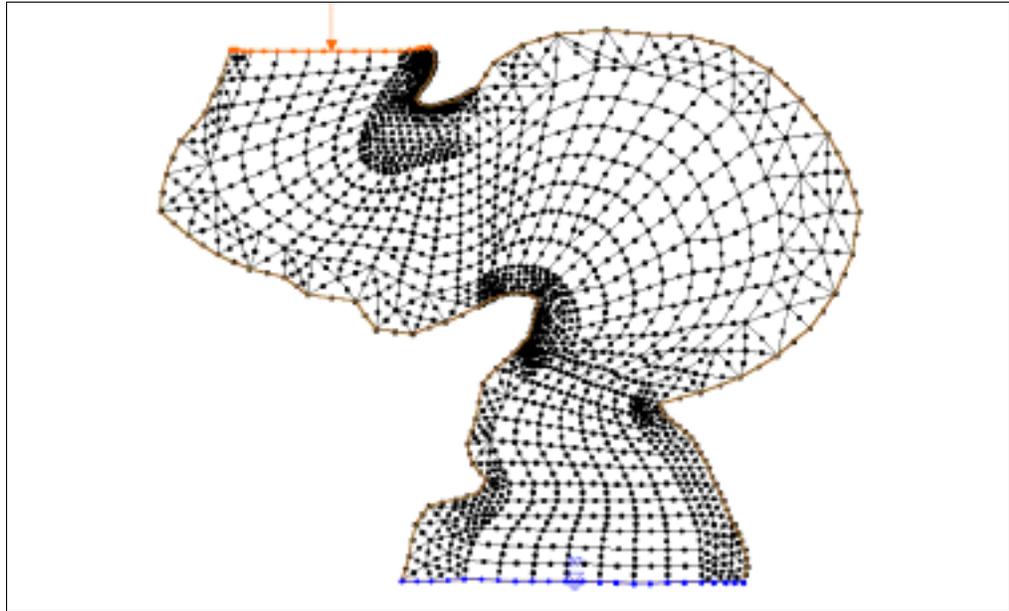


Figure 5-1. The mesh contained in *stmaryout.fil*.

## 5.2 Converting Elements

---

For *FESWMS*, it is best to use 9-noded quadrilateral elements (quads) even though both 8-noded and 9-noded quads are supported. The mesh contained in *stmaryout.fil* contains 8-noded quads. To convert these to 9-noded quads:

- Select *Elements* / *QUAD8<->QUAD9*.

The screen will refresh and the quadrilateral elements will have 9 nodes. Since there was a change in the number of nodes, the mesh should be renumbered, even though it was renumbered before being saved. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Click in the icon at the downstream boundary condition.
3. Select *Elements* / *Renumber* and click the *OK* button.

## 5.3 Defining Material Properties

---

Each element in the mesh is assigned a material type ID. After reading the file, *SMS* creates the materials with default values which must be changed for this simulation. To change the material values:

1. Select *FESWMS | Material Properties*. In the upper right of the *FESWMS Material Properties* dialog, an image shows what the Manning's coefficients are for different depths.
2. Highlight the material named *Material 1* and enter the following values:
  - 0.025 for both *Manning coefficients (n1 and n2)*.
  - 2.0 for *Depth 1* and 3.0 for *Depth 2*.
  - 20.0 for *Vo* and 0.6 for *Cu1*.
3. Highlight the material labeled *Material 2* and enter the same values that were entered for *Material 1* except enter 0.03 for Manning's roughness.
4. Highlight the material labeled *Material 3* and enter the same values that were entered for *Material 2*.
5. Click the *Close* button to close the *FESWMS Material Properties* dialog.

The eddy viscosity and Manning's roughness values should always be set. Other material properties can also be set for more advanced problems. See the *FESWMS* documentation for more information on these other material properties.

Optional: The materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* option. If you do this, be sure to turn the option back off before continuing with this lesson.

## 5.4 Setting Model Controls

---

Before running an analysis with this mesh, certain model controls and parameters must be set. The parameters and files used are specified in the *FESWMS Control* dialog. To change the global parameters:

1. Select *FESWMS | FESWMS Control*.
2. In the *FLO2DH Input* section, turn on the *NET File* option and turn off all the other options.
3. In the *FESWMS Version* section, choose *FESWMS 2.\**.
4. Make sure the *Units* are *English*, and the *Solution Type* is *Steady state*.
5. Click the *Parameters* button and set the values as shown in Figure 5-2. Then click the *OK* button to return to the *FESWMS Control* dialog.

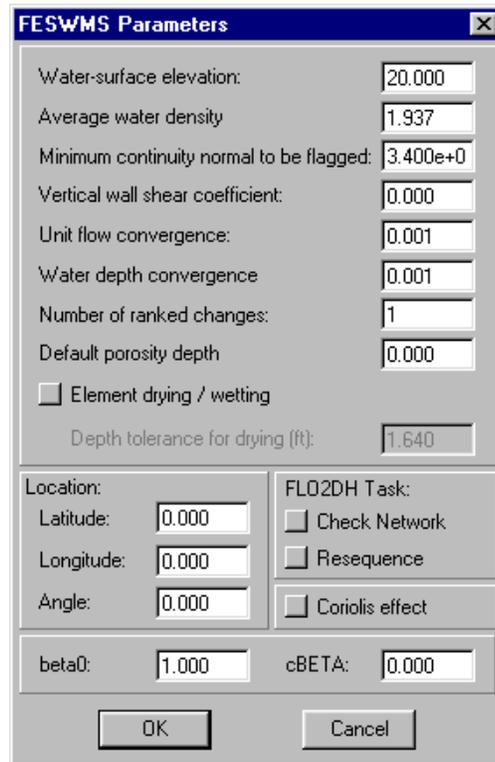


Figure 5-2 The FESWMS Parameters dialog.

6. Click the *Iterations* button and set the number of *Iterations* to 5. Then click the *OK* button to return to the *FESWMS Control* dialog.
7. Click the *Print* button and make sure the *ECHO to screen* option is turned on. Then click the *OK* button to return to the *FESWMS Control* dialog.
8. Click the *OK* button to exit the *FESWMS Control* dialog.

## 5.5 Model Check

Before running an analysis, the model should be checked for completeness. *SMS* provides a model checker for each numerical model that it supports. Although passing the model checker will not guarantee that the model will converge, some of the more common mistakes will be reported. To run the model checker:

1. Select *FESWMS / Model Check*.
2. Click the *Run Check* button.

The model checker should not report any warnings.

- Click the *Done* button to exit the *FESWMS Model Checker*.

## 5.6 Saving The Simulation

---

Now that model control parameters and material properties have been defined, and the model has been checked, the simulation is ready to be saved. To do this:

1. Select *FESWMS | Save Simulation*.
2. Save the simulation as enter *stmary3*.

The model control options and boundary conditions are saved to the file *stmary3.dat*, and the finite element network is saved to the file *stmary3.net*. If desired, look at the file *stmary3.fil* to see these filenames.

## 5.7 Using FLO2DH

---

You are now ready to run the analysis. The analysis module of *FESWMS* is called *FLO2DH* and it can be launched from inside *SMS*. To launch the *FLO2DH* program:

1. Select *FESWMS / Run FLO2DH*.
2. If the prompt shows a message that *FLO2DH* is *not found*, click the *File Browser*  button manually find the correct program executable.
3. Click the *OK* button to launch *FLO2DH*.

A new window will open in which *FLO2DH* will run the simulation. *FLO2DH* may take a few minutes to run, depending on the speed of your computer. When it is done, you will be prompted to press the *RETURN* key. If you are not prompted to do this, and the window goes away, then *FLO2DH* encountered an internal error and crashed. In this case, you may want to contact technical support with your files.

## 5.8 Conclusion

---

This concludes the Basic FESWMS Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## 2D Post Processing

---

### 6.1 Introduction

---

The solutions computed by the finite element analysis codes can be viewed in *SMS*. This is called *post-processing* of the finite element models. In this lesson, you will learn how to import, manipulate, and view solution data. You will need the geometry file *ld.2dm* and the solution file *ld0dyn.sol* created by a dynamic *RMA2* simulation.

---

### 6.2 Data Sets

---

When *SMS* imports a solution file, it creates a *data sets* for each quantity in the file. A data set contains solution data at each node of the mesh. *RMA2*, *FESWMS*, and *HIVEL2D* solutions create data sets representing velocity, water surface elevation, and water depth. *FESWMS* can create a special file with additional data sets for shear stress, Froude number, and energy head. An *RMA4* solution creates a data set for the concentration of each constituent. A *SED2D-WES* solution creates data sets representing sediment concentration and change in bed elevation. *ADCIRC* and *CGWAVE* solutions create data sets representing phase, angle, and amplitude of waves. *ADCIRC* also creates water surface elevation and velocity data sets. For steady state simulations, each data set only has a single time step, while for dynamic simulations, each data set has multiple time steps.

## 6.3 Using The Data Browser

All solution files are imported into *SMS* through the *Data Browser* dialog. Solutions may only be imported after the corresponding mesh has been opened. Furthermore, if the mesh is edited so that it no longer corresponds to a solution file, *SMS* will not open the solution file, and it will close any solutions that have already been opened. To open the finite element network for this example:

1. Select *File / Open*.
2. Open the file *ld.2dm* from the *tutorial* directory.

With the geometry opened, the solution can be imported. To import the solution file:

1. Select *Data / Data Browser* or click the *Data Browser*  macro.
2. Click the *Import* button.
3. *Import Data Set* dialog, set the *File Type* to *TABS file*, change the *Suffix* to “dyn”, and click the *OK* button.
4. Open the file *ld0dyn.sol*. This solution has 57 time steps in it. By default, the last time step of the velocity data set is highlighted.
5. Click the *Done* button to exit the *Data Browser* dialog.

When importing solution files, it is important to tell *SMS* which solution type is being opened. Table 6-1 shows which option to use for the different models.

Table 6-1 File formats for importing simulations.

| File Type            | Files Supported                 |
|----------------------|---------------------------------|
| Generic file         | HIVEL2D, SMS-created data files |
| TABS file            | RMA2, RMA4, SED2D-WES, RMA10    |
| FESWMS file          | FLOMOD, FLO2DH                  |
| ADCIRC file          | ADCIRC hydrodynamics            |
| ADCIRC harmonic file | ADCIRC wave information         |
| CGWAVE file          | CGWAVE                          |

After closing the *Data Browser* dialog, the display will refresh. If contours or vectors are being displayed, the new display will correspond to the data from the solution file. See the *SMS Reference Manual* for information on changing the display of contours and vectors.

## 6.4 Creating New Data Sets With The Data Calculator

*SMS* has a powerful tool, called the *Data Calculator*, for computing new data sets by performing operations on scalar values and existing data sets. In this example, a data set will be created, which contains the Froude number at each node. The Froude number is given by equation:

$$\text{Froude Number} = \frac{\text{Velocity Magnitude}}{\sqrt{\text{gravity} * \text{water depth}}}$$

To create the Froude number data set:

1. Select *Data / Data Calculator*.
2. Under the *Time Steps* section, turn on the *Use all time steps* option. This will compute the new function for each time of the dynamic simulation.
3. Highlight the *velocity mag dyn* data set and click the *Add to Expression* button. The expression will contain the letter corresponding to the *velocity mag dyn* data set.
4. Click the *divide* “ / “ button.
5. Click the *sqrt(x)* operation. Select the “??” text and delete it. This is just a place holder to make sure you know that something should be placed there.
6. Type the gravity value for English units, 32.17405.
7. Click the *multiply* “ \* “ button, then highlight the *water depth dyn* data set and click the *Add to Expression* button.
8. Click the *closing parenthesis* “ ) “ button.
9. The expression should now read: “c:all / sqrt(32.17405 \* b:all)”, where ‘c’ is the letter representing the velocity data set and ‘b’ is the letter representing the water depth data set.
10. In the *Result* field, enter the name *Froude*, then click the *Compute* button. *SMS* will take a few moments to perform the computations. When it is done, the *Froude* data set will appear in the *Data Sets* window.
11. Click the *Done* button to exit the *Data Calculator* dialog.

The display will refresh, showing contours of the Froude number data set. It can be treated just as any other dynamic scalar data set and can be saved in a generic data set file. See the *SMS Reference Manual* for more information on saving data sets.

## 6.5 Creating Animations

---

A film loop is an animation created by *SMS* to display changes in data sets through time. Flow trace animations are a special type of film loop which use vector data sets to trace the path that particles of water will follow through the flow system. Only the visible portion of the mesh will be included in the film loop when it is created. For this tutorial, zoom into the portion of the mesh shown in Figure 6-1. To do this:

1. Select the *Zoom*  tool from the *Toolbox*.
2. Click and drag the mouse to create a box about the area in Figure 6-1.

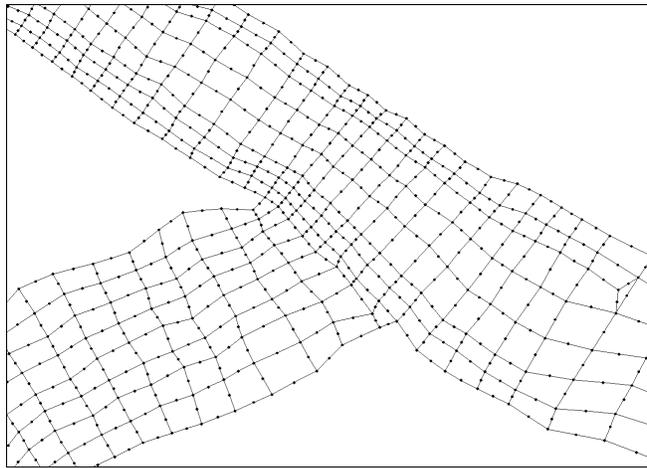


Figure 6-1. The area about which to zoom for the film loops.

### 6.5.1 Creating a Film Loop Animation

---

The film loop you to create will show how the velocity changes through time. To create and run the film loop:

1. Select *Data / Film Loop* and click the *Setup* button.
2. Click the *Data Browser*  button and make sure the *velocity mag dyn* and *velocity dyn* data sets are highlighted. Click the *Done* button.
3. Click the *Display Options*  button and turn off everything except *Velocity vectors*, *contours*, and the *Mesh boundary*.
4. Click the *Options* button next to *Velocity vectors* and choose the *Define min and max length* option with a *Min* of 10 and *Max* of 50. Also, choose the *Show normalized grid* option with a spacing of 30 in both directions.

5. Click the *OK* button to close both the *Vector Options* and *Mesh Display Options* dialogs, still leaving the *Film Loop Options* dialog open.
6. Increase the *Size (% screen)* value to 60%.
7. Make sure both the *Scalar data set* and *Vector data set* options are turned on, and make sure all time steps are selected (this should be the default).
8. Click the *OK* button in the *Film Loop Options* dialog to create the film loop.

*SMS* will display each frame of the film loop as it is being created, and a prompt in the *Edit Window* shows the frame being created. When the film loop has been fully generated, it can be played back using the following controls:

- *Play*  *button*. This starts the playback animation. During the animation, the speed and play mode can be changed.
- *Speed*. This increases or decreases the playback speed. The speed depends on your computer.
- *Stop*  *button*. This stops the playback animation.
- *Step*  *button*. This allows you to manually step to the next frame. It only works when the animation is stopped.
- *Loop*  *play mode*. This play mode restarts the animation when the end of the film loop has been reached.
- *Back/forth*  *play mode*. This play mode shows the film loop in reverse order when the end of the film loop has been reached.

When you are finished viewing the film loop, you may save it if *SMS* has been registered. A film loop created on the PC will be saved in the *AVI* file format. A film loop created on UNIX can be converted to the *AVI* file format using a free utility program which can be downloaded from the *SMS* web pages. *AVI* files can be used in software presentation packages, such as Microsoft PowerPoint or WordPerfect Presentations. In either case, a saved film loop may be opened and played in a later session. See the *SMS Reference Manual* for more information about viewing saved film loops.

### 6.5.2 Creating a Flow Trace Animation

A flow trace film loop can be created if a vector data set has been opened. The flow trace randomly introduces massless particles into the network and follows them through the vector field. Steady state vector fields can be used in a flow trace animation to show flow direction trends. For dynamic vector fields, the flow trace

animation can trace a single time step, or it can trace the changing flow field. Note that a flow trace will take upwards of 5 to 15 minutes to generate, depending on the speed of your computer. To create and run a flow trace film loop:

1. Once again click the *Setup* button from the *Film Loop* dialog.
2. In the middle left of the *Film Loop Options* dialog, turn on the *Flow Trace* option.
3. Select the *Use specified time steps* option and leave all the time steps of the velocity vector data set selected.
4. This time, use 30% for the screen size because flow trace animations take more memory to generate than film loop animations.
5. Click the *OK* button in the *Film Loop Options* dialog to create the flow trace.

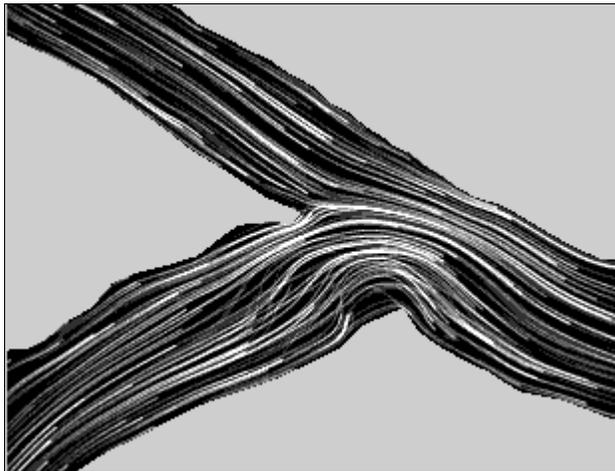


Figure 6-2. One frame from the *Id1* flow trace.

After a few moments, the first frame of the flow trace animation will appear on the screen. As before, the frames are generated one at a time, and a prompt shows which frame is being created. When the flow trace has been created:

- View the flow trace using the same controls as with the film loop.
- Save it if you wish and click the *Done* button in the *Film Loop* dialog.

## 6.6 Plots

---

Plots can be created to help visualize the data. Plots are created using the observation coverage in the map module. There is a special tutorial on the observation coverage which shows how to create data plots in lesson 16.

## 6.7 Conclusion

---

This concludes the 2D Post Processing tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## Advanced RMA2 Analysis

---

### 7.1 Introduction

---

This lesson will teach you how to use revision cards for a spin down simulation in *RMA2*. The geometry has already been created and renumbered. To open the file:

1. Select *File / Open*.
2. Open the file *ld.2dm* from the *tutorial* directory. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The geometry data will open, as shown in Figure 16-1.

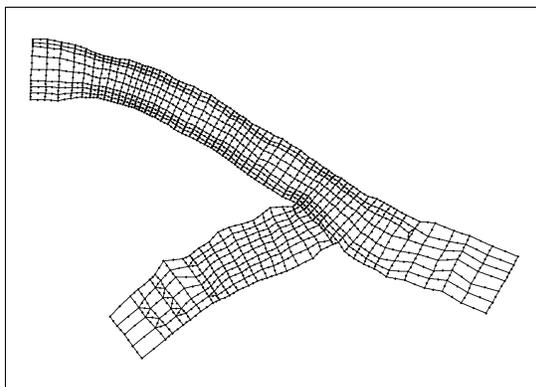


Figure 7-1. The mesh contained in the file *ld.2dm*.

## 7.2 Defining Material Properties

---

Each element in the mesh is assigned a material type ID. The materials were created with default parameters which must be changed for this particular simulation. The material properties define how water flows through the element (see the *SMS Reference Manual* for more information). To edit the material parameters:

1. Select *RMA2 / Material Properties*.
2. In the *RMA2 Materials Properties* dialog, highlight *Material 01*.
3. Make sure the *Isotropic eddy viscosities* option is on and enter a value of 25 for the eddy viscosity ( $E$ ).
4. Enter 0.03 for Manning's roughness ( $n$ ).
5. Highlight *Material 02* and set  $E$  to 50 and  $n$  to 0.04.
6. Click the *Close* button to close the *RMA2 Material Properties* dialog.

The material parameters have now been defined. You may turn on the display of the materials if you want. To do this, open the *Display Options* dialog and turn on the *Materials* option. If you do this, turn them back off before continuing.

## 7.3 Creating Nodestrings

---

For this tutorial, flow and water surface elevation will be defined along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where flow enters or exits from the mesh. Generally for *RMA2*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries. Three boundary strings must be created, one at each of the open boundaries. These boundaries are highlighted in Figure 7-2. To create a nodestring across the upper left boundary:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the upper left corner node.
3. Hold the *SHIFT* key and click on the bottom node of the inflow boundary. *SMS* will automatically select all nodes between the two.
4. Press the *ENTER* key to terminate the nodestring.

In a similar fashion, create nodestrings at the other two open boundaries.

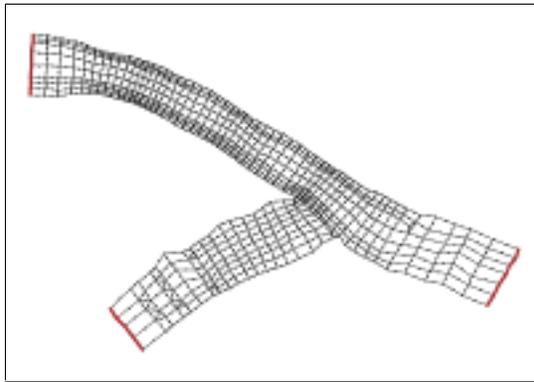


Figure 7-2. Position of the boundary nodestrings in the mesh.

## 7.4 Defining a Dynamic Simulation

The finite element network and associated material properties are only part of the numerical model. In addition, several other model control parameters must be defined. These model parameters include items such as how to handle wetting and drying, the model units, simulation time, and how many iterations should be made to get the solution to converge. Information on each parameter can be found in the *SMS Reference Manual* and the *RMA2* documentation.

For this simulation, a very low water surface will be defined. However, to be able to assign the water surface value, a hotstart file will be generated. The desired head value is so low that a few elements may become dry, but the marsh porosity option will be used. To define the model parameters:

- Select *RMA2 / Model Control*.

This opens the *RMA2 Model Control* dialog in which various parameters will be set in the next few sections.

### Defining Model Units

- Change the *Units* to *English Units*.

### Defining Time Parameters

The first simulation should be a dynamic simulation with 57 time steps. To do this:

1. Change the *Solution Type* (bottom right) to *Dynamic*.
2. In the *Computation Time* section, use the *Specify time step time* option with a *Time step* of 10.0 hours, and then 57 for the *Number of time steps*.

### Defining Hotstart Output

This simulation will be used to create a hotstart file for a steady state simulation. To do this:

- Turn on the *Hotstart output file* option (middle left).

### Defining Iteration Controls

Since the water surface for this model will become rather low, and thus unstable, the number of iterations should be increased and a convergence parameter should be defined to make sure the model will converge. To do this:

1. Set both the *Initial solution* and *Each time step* iterations to 20.
2. Turn on both the *Steady state depth convergence* and the *Dynamic depth convergence* options, and enter a value of 0.001 for each.

Each time step of the model will be considered converged when the maximum change in water depth at all nodes is less than 0.001 ft.

### Defining Wetting/Drying Parameters

Wetting/drying parameters are not required, so they are in the *Optional BC Controls* dialog. To set these parameters:

1. Click the *Optional BC Controls* button.
2. Turn on the *Dry Elements* option.
3. Set the *Check wet/dry testing* iterations to 4, the *Nodal dry depth* to 0.001 ft, and the *Nodal active depth* to 0.10 ft.
4. Turn on the *Marsh Porosity* option and click the *Options* button.
5. Set *AC1* to 4.0, *AC2* to 0.1, and *AC3* to 0.001, then click the *OK* button.

To accept all the above values:

- Click the *OK* button to close both model control dialogs.

---

## 7.5 Defining Transient Boundary Conditions

There are various uses for transient boundary conditions, such as to model a tide (transient head) or run a storm hydrograph through a river (transient flow). This example uses transient boundary conditions to create a hotstart file for increasing the model stability.

The water surface elevation will be spun down to a desired starting condition so that a steady state model can be run, while the flow conditions will be constant. The head curve will be imported from a file. (See the *SMS Reference Manual* for information on creating time series curves.)

### 7.5.1 Defining Flow Boundary Conditions

---

To assign the flow conditions:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the top left nodestring by clicking in its icon.
3. Select *RMA2 / Assign BC*.
4. In the *RMA2 Assign Boundary Conditions* dialog, set the *Condition Type* to *Flow BC* and assign a flow of 55000 (cfs).
5. Make sure the *Flow direction* is set to be *Perpendicular to Boundary*.
6. Click the *OK* button to exit the *RMA2 Assign Boundary Conditions* dialog.

This defines the left nodestring to be an inflow boundary condition. To define the second inflow boundary condition:

1. Select the bottom left nodestring.
2. Assign a perpendicular flow of 580 cfs.

### 7.5.2 Defining Head Boundary Conditions

---

The head boundary condition will be a transient curve and will be imported from a file. To assign this boundary condition:

1. Select the nodestring on the right boundary.
2. Select *RMA2 / Assign BC*. Make sure the *Head BC* option is selected.
3. Change the boundary condition type to *Transient* and then click the *Define Curve* button.
4. In the *XY Series Editor* dialog, click the *Import* button and open the file *head\_ld.xls*. A transient curve will be displayed in the window.
5. Click the *OK* button in both dialogs to assign the boundary condition.

## 7.6 Running The Model Checker

---

Before running the analysis, it is a good idea to run the model checker. The model checker looks for potential problems and common mistakes in the mesh. To run the model checker:

1. Select *RMA2 / Model Check*.
2. In the *RMA2 Model Checker* dialog, click the *Run Check* button.

The model checker should come up with only one warning. This warning says that the mesh has not been renumbered since it was opened. This can be ignored because, as stated earlier, the mesh was already renumbered in an earlier session. Although the model checker does not come up with any warnings, this does not mean that the model will converge. However, it means that there should not be any serious errors. Although the *RMA2 Model Checker* dialog can be open while using *SMS*, it will not be needed anymore. To close it:

- Click the *Done* button to close the *RMA2 Model Checker* dialog.

## 7.7 Saving The Simulation

---

Now that the boundary conditions and global parameters have been defined and checked, the simulation can be saved. The *TABS* programs have incorporated a way to be launched from inside *SMS* after a simulation is saved. To save the simulation:

1. Select *RMA2 / Save Simulation*.
2. Change the *Simulation file* name to *ld0* and click the *Update* button.
3. Click the *OK* button to save the simulation.

## 7.8 Using Revision Records

---

The simulation that has been saved is a dynamic simulation which will produce a hotstart file. This hotstart file will be used as input to a steady state model. However, a dynamic hotstart file cannot be used as input into a steady state model. Because of this, a change needs to be made to the boundary condition file to convert the time steps into revisions. Using revisions causes *RMA2* to see a steady state model, which can be used to hotstart a new steady state model. To make this change to the boundary condition file:

1. Select *Display / View Data File*.

2. Change the *Files of type* filter to *TABS BC (\*.bc)* and open the file *ld0.bc*.
3. If you want to use a text editor different from the default, enter the text editor name and click the *OK* button.
4. Change each *END* record to *REV*, except for the last one. The file will look like this:

```
REV Simulation at time = 0.00
.
.
REV Simulation at time = 10.00
.
.
END Simulation at time = 570.00
STOP
```
5. Save the file and close the text editor.

The simulation can now be run through *RMA2*.

---

## 7.9 Running The Initial Simulation

---

### 7.9.1 Running GFGEN

---

Before running the finite element analysis, the ASCII geometry file must be converted to a binary file that *RMA2* can understand. This is done with a program called *GFGEN*. To launch the *GFGEN* program:

1. Select *RMA2 / Run GFGEN*. A prompt appears, showing the location of the most recent *GFGEN* executable.
2. If a message such as “gfgv435.exe – not found” is given, then click the *File Browser*  button to manually find the *GFGEN* executable.
3. Click the *OK* button to launch *GFGEN*.

When the *GFGEN* window finishes, you will get a message to press the *RETURN* key. If you do not get such a message and the window goes away, then *GFGEN* crashed and you may want to contact technical support with your geometry file.

### 7.9.2 Running RMA2

---

After *GFGEN* is successfully completed, the finite element analysis can run. To launch the *RMA2* program:

1. Select *RMA2 / Run RMA2*. As with *GFGEN*, the prompt shows the location of the most recent *RMA2* executable.
2. If necessary, find the executable, and click the *OK* button to launch *RMA2*.

Once again, a window will appear, this time running *RMA2*. After the model runs, you should be prompted to press the *RETURN* key. When the solution has been finished, a file named *ld0\_out.hot* will be created, which can be used as an initial condition file for a steady state simulation.

## 7.10 Defining a Steady State Simulation

---

Now that the hotstart file has been generated, the steady state simulation can be created. Most of the model parameters will remain the same as those from the “dynamic” simulation, such as wetting/drying and model units. There are only a few changes that need to be made. To make these changes:

1. Select *RMA2 / Model Control*.
2. Change the *Solution Type* to *Steady State*. Notice that the *Time Control* and *Dynamic Depth Convergence* sections become unavailable.
3. Turn on the *Hotstart input file* option to have *RMA2* look for a hotstart file to use as the initial conditions.
4. Click the *OK* button to close the *RMA2 Model Control* dialog.

## 7.11 Defining Constant Boundary Conditions

---

Constant boundary conditions, as the name implies, do not change with time. The inflow boundaries were initially defined as constant boundary conditions. For the steady state simulation, these will remain the same. The head boundary needs to be changed to a constant condition. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the right nodestring (head boundary) by clicking in its icon.
3. Select *RMA2 / Assign BC*.
4. In the *RMA2 Assign Boundary Conditions* dialog, the boundary condition is automatically changed to Constant. Enter a value of 40.0 (feet).
5. Click the *OK* button to close the *RMA2 Assign Boundary Conditions* dialog.

The definition of the steady state simulation is now complete.

## 7.12 Saving The New Simulation

---

This simulation needs to be saved. To do this:

1. Select *RMA2 / Save Simulation*.
2. Turn off the *GFGEN Files* option since no change was made to the mesh.
3. In the *Simulation file* field, enter *ld1* and click the *Update* button.
4. Make sure the *Binary geometry* filename is still *ld0*.
5. Change the *Hotstart input* filename to *ld0\_out*.
6. Click the *OK* button to save the simulation.

This saves the new steady state simulation so that *RMA2* can be run.

## 7.13 Running The Final Simulation

---

After saving this simulation, *RMA2* can be run to generate the final solution. To launch the *RMA2* program:

1. Select *RMA2 / Run RMA2*.
2. Click the *OK* button to the prompt.

Once again, a window will open while *RMA2* runs. The file *ld1.sol* is the final solution file, and contains the steady state flow and head data at each node.

## 7.14 Conclusion

---

This concludes the Advanced RMA2 Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## ***Advanced FESWMS Analysis***

---

### **8.1 Introduction**

This lesson will teach you how to prepare an advanced *FLO2DH* simulation, including the use of weirs. First, make sure you are in the correct portion of *SMS*. To do this:

1. In the *Mesh*  module, select *Data / Switch Current Model*.
2. Select the *FESWMS* model. If it is unavailable, you will need to run *SMS* in *Demo Mode* (see section 1.4).
3. Click the *OK* button.

---

### **8.2 Opening The Geometry**

You will be using the file *suecreek.fil* as shown in Figure 8-1. This mesh has been created and renumbered in a previous session. Although *FLO2DH* supports both 8- and 9- noded quadrilateral elements, this mesh contains only 9-noded quadrilaterals. To open the mesh data:

1. Select *FESWMS / Open Simulation*.

2. Highlight the file *suecreek.fil* in the *tutorial* directory and click the *Open* button. If a mesh is already open, you will be warned that the previous mesh will be deleted. If this happens, click the *OK* button at the prompt.

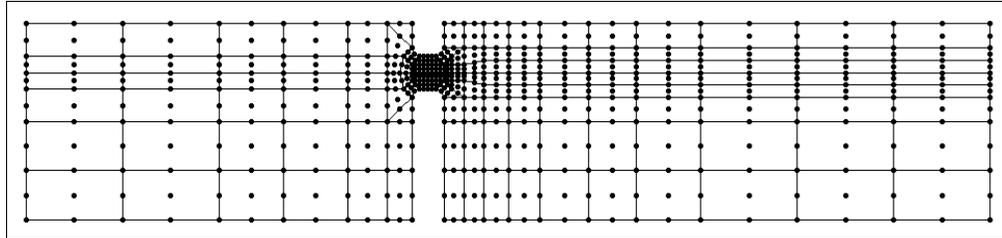


Figure 8-1. The *suecreek.fil* geometry.

### 8.3 Defining Material Properties

---

Each element of the mesh is assigned a material ID. The material ID tells *FLO2DH* which material properties should be assigned to the element. There are four different material types in this mesh, but the material properties have not been defined. When *SMS* opens a mesh with undefined materials, the materials are assigned default properties. See the *FESWMS* documentation for a definition of individual material parameters. To change the material values:

1. Select *FESWMS / Material Properties*. A graphical image in the upper right corner of the *FESWMS Material Properties* dialog shows the Manning's *n* value as a function of water depth.
2. Highlight *Material 1*, and be sure that its *ID* is 1.
3. Enter the following values:
  - 0.035 for both *n1* and *n2*
  - 2.0 for *Depth 1* and 3.0 for *Depth 2*
  - 20.0 for *Vo*
  - 0.6 for *Cu1*
4. Highlight *Material 2* and check that its *ID* is 2.
5. Enter the same values for *Material 2* that were entered for *Material 1*.
6. For *Material 3* and *Material 4*, enter the same values as above, except that *n1* and *n2* are both 0.055.

7. Click the *Close* button to accept these changes and close the *FESWMS Material Properties* dialog..

You have just assigned values for the four materials in this mesh. Notice that there are only two distinct material regions because materials 1 and 2 have the same values, as do materials 3 and 4.

Optional: The materials can be displayed by opening the *Mesh Display Options* dialog and turning on the *Materials* option. See the *SMS Reference Manual* for more about changing the display of materials. You may turn on the display of materials, but if you do so, be sure to turn them off before continuing with this tutorial.

## 8.4 Creating The Hotstart File

---

We wish to model this portion of river with 9,000 cfs of flow. However, this model will not converge when using such a large initial flow rate. An intermediate solution file using a lower flow rate will first be created. This intermediate solution will then be used as a hotstart file so that a solution with the desired flow can be computed.

### 8.4.1 Assigning Boundary Conditions

---

Boundary conditions such as flow and head define how water enters and leaves the finite element network. Without proper boundary conditions, instability of the model and inaccuracy of the solution will result.

#### Defining Flow and Head

A steady state model such as this can only have constant boundary conditions. The flow and head boundary conditions will be defined at nodestrings on opposite sides of the model as shown in Figure 8-2. To create the two boundary nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. Click on the upper node of the left boundary.
3. Hold the *SHIFT* key and click on the lower node of the left boundary. This creates the nodestring all the way across the left boundary of the mesh. If you did not hold the *SHIFT* key, it would not be a valid boundary nodestring because it would not include all nodes across the boundary.
4. End the nodestring by pressing the *ENTER* key.
5. Repeat this procedure to create a second nodestring on the right boundary.

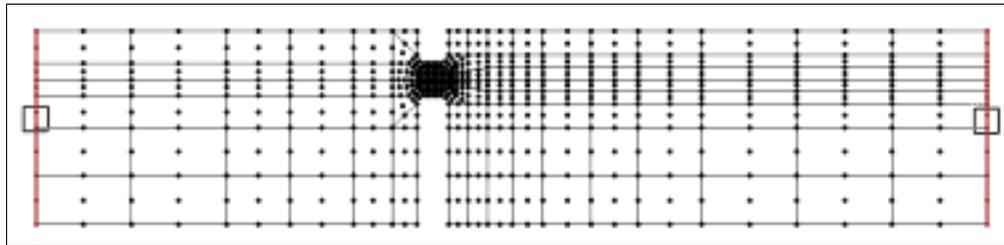


Figure 8-2. Location of the boundary condition nodestrings.

Boundary conditions can now be assigned to the nodestrings. To assign the flow to the left boundary:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring, as shown in Figure 8-2.
2. Select the nodestring on the left boundary by clicking inside its icon.
3. Select *FESWMS / Assign BC*.
4. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Flow* option and assign a value of 5000 (cfs). Be sure that the *Normal* option is selected.
5. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a flow nodestring and its color changes. An arrow appears at the center of the nodestring to indicate the flow direction and the flow value is shown next to the arrow (see Figure 8-3).

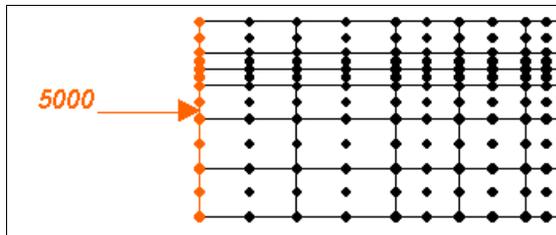


Figure 8-3 The inflow boundary condition.

To assign the head to the right boundary:

1. Select the right nodestring.
2. Select *FESWMS / Assign BC*.
3. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Water surface elevation* option and assign a value of 812.9 (feet). Be sure that the *Essential* option is selected.

- Click the *OK* button to close the dialog.

The selected nodestring is now defined as a head nodestring and its color changes. A head symbol appears at the center of the nodestring and the head value is shown next to the symbol (see Figure 8-4).

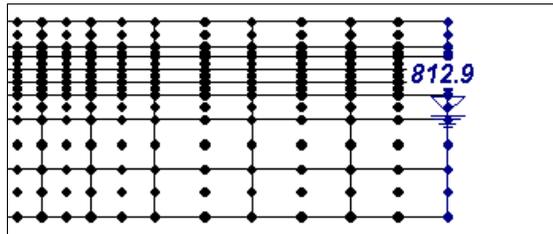


Figure 8-4 The outflow boundary condition.

## 8.4.2 Creating Weirs

With *FESWMS*, flow control structures such as weirs, piers, culverts, and drop inlets are easily added to the mesh. Weirs, culverts, and drop inlets are created between pairs of nodes. Wide structures can be created between strings of node pairs. For this model, a weir will be defined along five node pairs across the abutment at the bottom middle of the mesh, as shown in Figure 8-5.

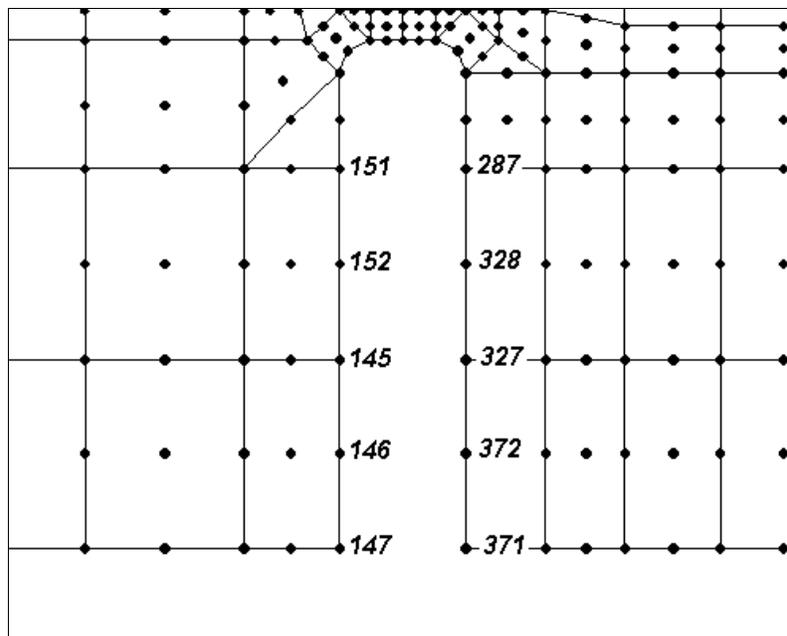


Figure 8-5 Area where weirs will be added.

Turn on the display of node numbers using the *Mesh Display Options* dialog. Zoom in on the lower portion of the middle of the bridge as shown in Figure 8-5. The five

node pairs for the weir are: 151<->287, 152<->328, 145<->327, 146<->372, and 147<->371. To select the first pair of nodes:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. While holding the *SHIFT* key, click on nodes 151 and 287.

To create a weir segment between the selected nodes:

1. Select *FESWMS / Weir*.
2. Make sure the *upstream* node is 151, and the *downstream* node 287. If these are opposite, click the *Switch* button.
3. Enter the following values:
  - 0.53 for the *Discharge Coefficient*.
  - 25 for *Crest Length*.
  - 825 for the *Crest Elevation*.
4. Click the *OK* button or press the ENTER key.

The first weir section has now been defined. The last four weir sections are defined in a similar manner. To finish the weir:

- Select each pair of nodes and assign weir values as shown in Table 8-1.

Table 8-1 The weir node pair data.

| Upstream | Downstream | Discharge | Length | Elevation |
|----------|------------|-----------|--------|-----------|
| 151      | 287        | 0.53      | 25     | 825       |
| 152      | 328        | 0.53      | 100    | 825       |
| 145      | 327        | 0.53      | 50     | 825       |
| 146      | 372        | 0.53      | 100    | 825       |
| 147      | 371        | 0.53      | 25     | 825       |

When these values are all entered, you will have defined a single weir which spans the two bottom elements. This weir is 300 feet long, has an elevation of 825 feet, and has a discharge coefficient of 0.53. There is a specific formula for determining the crest length of each pair of nodes. Each midside node has 2/3 of the element width in its crest length while each corner node has 1/6 of each element width that is involved in the weir. In this case, there are two elements involved in the weir, both of which are 150 feet long. This yields the distribution given in the table. See the *SMS Reference Manual* for more information on weirs and other flow control structures.

After creating the weir, reset the display to the way it was before starting the weir creation, as shown in Figure 8-6. To do this:

1. Turn off the display of node numbers using the *Mesh Display Options* dialog.
2. Click the *Frame*  macro in the *Toolbox* to frame the image.

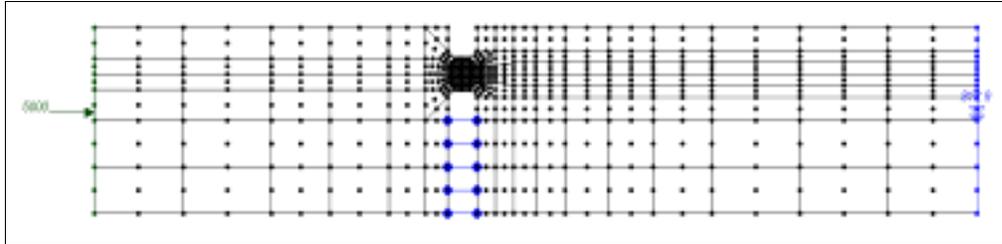


Figure 8-6. The final bridge geometry.

### 8.4.3 Saving The Data

With *FESWMS* software, data is saved in multiple files. The file names are specified in the *FESWMS Model Control* dialog. To set up the *FESWMS* file options:

1. Select *FESWMS / FESWMS Control*.
2. In the *FESWMS Version* area, choose *FESWMS version 2.\**.
3. In the *FLO2DH Input* section, turn off all options except for the *NET File*.
4. Click the *OK* button or press the *ENTER* key.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *FESWMS / Save Simulation*.
2. Enter the name *suecreek2.fil*.
3. Click the *Save* button.

The model control parameters and boundary conditions are saved to *suecreek2.dat* while the node and element information is saved to *suecreek2.net*. The main file is called *suecreek2.fil* and it contains a list of the filenames that were specified. The only input supplied to *FLO2DH* is this *suecreek2.fil* file, but *FLO2DH* uses all the files that were created.

### 8.4.4 Using FLO2DH

---

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FLO2DH*. To run *FLO2DH*:

1. Select *FESWMS / Run FLO2DH*.
2. If the prompt shows a message that *flo2dh* is *not found*, then click the *File Browser*  button and find the location manually.
3. Click the *OK* button to run *FLO2DH*.

A window will open to run the *suecreek* model through *FLO2DH*. Depending on the speed of your computer, *FLO2DH* will take a few minutes to finish the solution. Upon completion, *FLO2DH* writes a solution file named *suecreek2.flo*. This file contains the velocity and water surface elevation for each node in the mesh. The solution can be read into *SMS* using the *Data Browser* (see the *SMS Reference Manual* for more information).

Note: If you have not registered the *FESWMS* interface, you can still run the model in a prompt by using the supplied *suecreek2.\** files in the *tutorial/output* directory.

## 8.5 Reworking The Solution

---

As previously stated, the solution that was computed in section 8.4.4 is only an intermediate solution. It will be used as a hotstart file for an altered set of boundary conditions which will be set up in this section.

### 8.5.1 Changing The Boundary Conditions

---

The flow value needs to be increased to the full desired value. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the flow nodestring (left boundary) by clicking in the icon.
3. Select *FESWMS / Assign BC*.
4. Increase the flow value to 9000 (cfs).
5. Click the *OK* button

### 8.5.2 Editing The Weir Data

---

If you look at the previous solution data, you will see that there is no flow through the weirs. The water surface at the weirs is much less than the weir crest elevation, so overtopping did not occur. This was purposely done to add model stability. Now, however, the weirs need to be lowered to their true elevation of *812.5 ft*. To do this:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. Select one pair of nodes which makes up one of the weir sections. Each weir pair is shown connected by a line.
3. Select *FESWMS / Weir*.
4. Change the *Crest Elevation* to *812.5 feet*.

Repeat these steps to lower the crest elevation of all weir segments to 812.5 feet.

### 8.5.3 Using The Hot Start file

---

*SMS* needs to tell *FLO2DH* to use the previous solution file as an input *hotstart*, or *initial condition*, file. To do this:

1. Select *FESWMS / FESWMS Control*.
2. In the *FESWMS Control* dialog, turn on the *INI file* option.
3. Click the *File Browser*  button to the right of this option. Find and select the file *suecreek2.flo* that was created when *FLO2DH* ran. If you were not able to run the model, you can use the solution file that is provided in the *tutorial/solution* directory.
4. Click the *OK* button in both dialogs.

### 8.5.4 Computing a New Solution File Using a Hotstart File

---

To run the new simulation:

1. Using the steps in section 8.4.3, save the modified simulation file as *suecreek3.fil*. Be sure to save the new simulation in the same directory as the *suecreek2.flo* file or *FLO2DH* will not find *suecreek2.flo*.
2. Use the steps in section 8.4.4 to run a new simulation. A new solution file named *suecreek3.flo* will be created by *FLO2DH*.

The solution file can be imported into *SMS* through the *Data Browser* (see the *SMS Reference Manual* for more information). Once a solution file has been read, *SMS* processes the data sets created from them in a model independent fashion. Therefore the methods for post-processing described in Lesson 6 are applicable for *FESWMS* solution files.

## 8.6 Conclusion

---

This concludes the *Advanced FESWMS Analysis* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.

---

***SED2D-WES Analysis***

The SED2D-WES Analysis tutorial is coming soon!



---

## ***RMA4 Analysis***

---

### **10.1 Introduction**

This lesson will teach you how to run a solution using *RMA4*. If you have not yet completed Lesson 4 on *RMA2*, you should do so now. *RMA4* is part of the *TABS-MD* suite of programs and is used for tracking constituent flow in 2D models. This lesson will use a 2D mesh and a solution file that have already been generated.

---

### **10.2 Preparing For RMA4**

*RMA4* can only be run after having initially run a solution in *RMA2*. This is because *RMA4* uses the flow solutions computed by *RMA2* to compute the constituent concentration as it flows through the mesh.

Open the file *noyo.geo* as shown in Figure 10-1. To do this:

1. Select *RMA2 / Open Simulation*.
2. Select the file *noyo.geo*. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button.
3. In the prompt that appears, click the *Geometry* button.

The geometry will be displayed on the screen. The solution file has already been generated and is named *noyo.sol*.

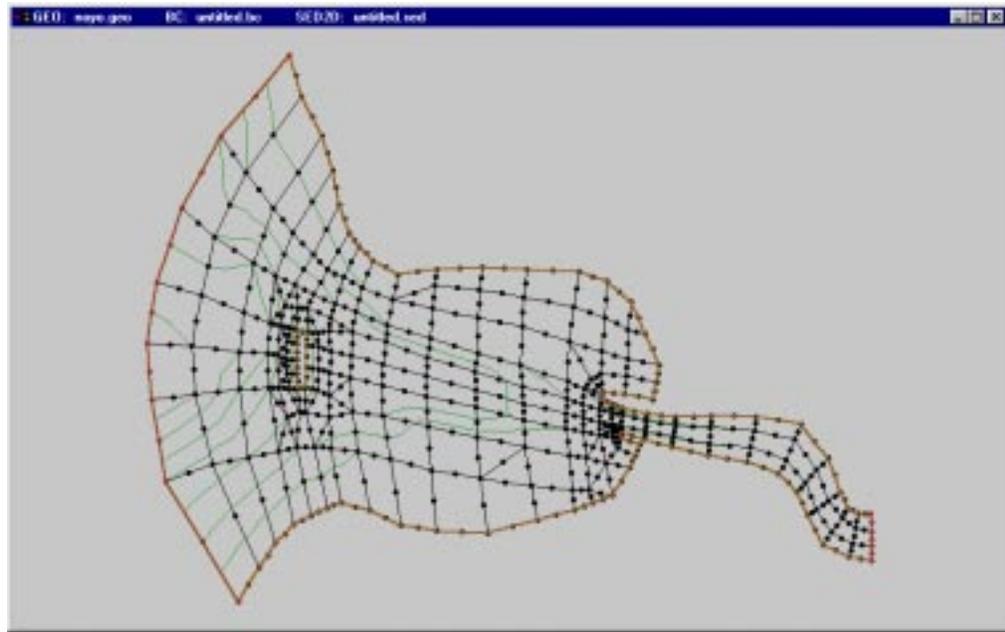


Figure 10-1 Noyo Bay mesh

## 10.3 Creating a RMA4 Input File

---

The information for an *RMA4* simulation is entered by creating a text file using a text editor such as *Wordpad* or *vi* for UNIX. Sample input files for different cases have been prepared. The cases provide sample files for:

- Constant point load
- Variable point load
- Salinity intrusion

### 10.3.1 Creating a Constant Point Load Input File

---

Open the file *noyo\_con.trn* using a text editor to follow along, or create a new text file by entering the values specified. The input file is organized using a *HEC* card format. The first 2 or 3 capital letters in each line define what information will be contained on that line. Here is a brief description of what each of the cards represent. Below are the values that we will be using in our file.

**T1, T2, T3-** Title. Any type of information can be entered in this line describing the run.

```
T1
T2 Noyo Bay test case by Rob Wallace
T3
```

**\$L-** File Identifier codes. (8 values) This line should always be defined using the values specified.

```
$L 10 20 0 0 31 0 33 0
```

**\$M-** Machine Identifier. (1 value) This value corresponds to the type of computer that you are using. A value of 1 is used for a PC. For other values consult the *RMA4* documentation.

```
$M 1
```

**GS HS-** Conversion Factors. (2 values) & (3 values) If English units are being used .3048 is entered as a conversion factor. If using Metric units these lines can be omitted.

```
GS .3048 .3048
HS .3048 .3048 .3048
```

**TC-** Time Control. (5 values) The values representing the time steps for the simulation are: start-time time-step total-steps max-time SSflag. The SSflag should be set to 0. The simulation will stop when either total-steps or max-time is reached.

```
TC 0.0 0.5 25 12.0 0
```

**TH-** Time Info. (2 values) The first is the number of hours to be subtracted from the *RMA2* input velocity file. The second is the last time step from the *RMA2* velocity file to be used.

```
TH -0.5 12.0
```

**TO-** Frequency to save. (2 values) The values are set for the saving of the output file: value1- beginning time step to start saving, a negative or 0 value causes all time steps to be saved; value2- time step to stop saving, or a negative or 0 value causes all time steps to be saved.

```
TO -1 -1
```

**TP-** Time Print Control. (5 values) The values are set to optimize printing: value1- (0) do not print node/element data, (1) print everything, (2) do not print node/element data and print short form of the results; value2- (0) do not print time steps, (+num) print time step intervals; value3- (0) do not echo inputs, (1) echo inputs; value4- (1) trace program logic during run, (0) do not; value5- (0) no detail internal print trace, (1-4) diagnostic debug print trace. Generally the values specified below should be used.

```
TP 1 1 0 0 0
```

**FQ-** Constituent Control. (2 values) The first is the number of constituents. The second tells what the constituent is (1) should be entered for dissolved Oxygen, (2) for BOD, (0) other.

```
FQ 1 0
```

**FQC-** Constituent Decay Coefficient. One value should be entered for each constituent. (0) for no decay, a number between 0 and 1 for growth, and infinity for rapid decay.

```
FQC 0.0
```

**DF-** Diffusion Coefficients. (3 values) An integer representing the material ID and two decimal values representing the x and y diffusion coefficients. There should be one DF card for each material.

```
DF 1 -1.0 -1.0
DF 2 -1.0 -1.0
```

**FT-** Temperature. (1 value) The water temperature in degrees Celsius. If the card doesn't exist 15.0 degrees Celsius will be used.

```
FT 17.0
```

**IC-** Initial Concentration. (2 values) The first should be the integer 1 and the second is a decimal value of the initial concentration of the constituent existing in the water.

```
IC 1 0.0
```

**BCN-** Nodal Boundary Condition. (4 values) The values are: value1- node number where the constituent will be introduced; value2- the concentration of the constituent being introduced; value3- (0) no factor will be applied if the flow changes direction, (1) a factor is applied; value4- a factor to be applied if the preceding value is 1, the factor is a decimal value between 0 and 1 with 0 representing shock and 1 a very gradual change.

```
BCN 639 8.0 0 0.0
```

**BCC-** Boundary Condition Type. (2 values) The first value is the time step in decimal hours. The second value is the number of time steps to be skipped off of the input file. Usually the time step will be equal to that set in the TC card and the second value will be zero.

```
BCC 0.5 0
```

**RE-** Re-solve Solution. (3 values) The values are: value1- (1) save the global matrix, (-1) do not, (0) use value2; value2- (n) use resolve file saved during the time step n, (0) use value1; value3- time step for this solution step (only used when value1 is active).

```
RE -1 0 0.5
```

**END-** End of Time Step. There should be as many *END* cards as there are time steps. After the *END* card any comment can be entered. If there are not as many *END* cards as there are time steps, the simulation will stop early.

```
END End of time step 1
END End of time step 2
"
END End of time step 25
```

**STOP-** This card should be at the very end of the file to stop the simulation.

```
STOP
```

With the information entered, the file is saved as a *.trn* file, which is the input format for *RMA4*. This file can now be used with the *RMA2* solution file to run *RMA4*.

### 10.3.2 Creating a Variable Point Load Input File

---

The file that was just created for the constant point load, can be modified to run a variable point load model. To modify the constant point load file:

1. Find the *BCN* card.
2. Change the second value on the *BCN* card, which represents the concentration of the constituent being introduced, from 8.0 to 0.0.
3. Count down two *END* cards and below the *End of time step 2* line create a new *BCN* card with the values:
 

```
BCN 639 8.0 0 0.0
```
4. Count down two more *END* cards and below the *End of time step 4* line create a new *BCN* card with the values:
 

```
BCN 639 0.0 0 0.0
```
5. Save the file as *noyo\_pt.trn*.

This input file will model a situation where there is originally no constituent present, after two time steps, the constituent is introduced. After two more time steps, the constituent is stopped. This model will simulate flushing the constituents out and the concentrations that are left.

### 10.3.3 Creating a Salinity Intrusion Input File

---

An input file to model salinity intrusion is similar to the input file for a constant point load. For this situation, we a nodestring will be defined in the mesh, and the constituent will be introduced along the nodestring. Open the file *noyo\_con.trn* with the text editor on your computer.

1. Find the *BCN* card.
2. Insert a *GC* card above the *BCN* card. The *GC* card is used to define a nodestring. The first value is a number to represent the node string. It is followed by the ID of each corner node in the nodestring. The card is terminated with a negative one. The left mesh boundary will be defined as nodestring one. Create the *GC* card as follows:
 

```
GC 1 7 12 17 22 27 32 37 -1
```
3. Now change the *BCN* card to a *BCL* card. The *BCL* card works the same as the *BCN* card, except that it specifies that the constituent will be introduced from a nodestring instead of a single node. Also change the node number

639 to nodestring number *I*. The rest of the values will be left the same. The string should now look like:

```
BCL 1 8.0 0 0.0
```

4. Save the file as *noyo\_sal.trn*.

## 10.4 USING RMA4

---

*RMA4* is an analysis program which computes a constituent concentration at each node. *RMA4* requires the binary geometry file created by *GFGEN*, the *RMA2* solution file and the ASCII initial condition file that was created in this tutorial.

To run *RMA4*:

1. In a terminal window, enter the path to the directory containing the initial condition file and the output from *RMA2*.
2. Type *rma4v431* to execute the program.
3. Respond to the prompts that appear as follows:

```
Enter run control input file name:  
Noyo_con.trn  
Enter full print output file name:  
Noyo_con.ot1  
Enter input geometry file (binary):  
Noyo.bin  
Enter velocity input file:  
Noyo.sol  
Enter final RMA-4 results file:  
noyo_con.sol
```

*RMA4* can take between five and ten minutes to run this solution, depending on the speed of your computer. When *RMA4* has completed the calculations, a beep will sound. The file *noyo\_con.sol* is the solution file containing constituent data at each node.

## 10.5 Conclusion

---

This concludes the *RMA4* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

Select the *File / Quit*. If prompted to confirm, click the *Yes* button.

---

## ***HIVEL Analysis***

---

### **11.1 Introduction**

---

This lesson will teach you how to prepare a mesh for and run a solution using *HIVEL2D*. You will be using the file *proto.2dm*, which is a finite element mesh.

Before opening the file, you should make sure *SMS* is using the *HIVEL2D* interface. To switch to the *HIVEL2D* interface:

1. Make sure you are in the *Mesh*  module.
2. Select *Data | Switch Current Model...*
3. Select the *HIVEL2D* option and press the *OK* button.

With *SMS* in the *HIVEL2D* interface, you are ready to open the geometry file. To do this:

1. Select *File | Import* and select the *2-D Mesh option*. Click *OK*.
2. Find and highlight the file *proto.2dm*. If you still have geometry data open from a previous tutorial, you will be warned that all existing mesh data will be deleted. If this happens, click *OK* to the prompt. The geometry data will open as shown in Figure 11.1.

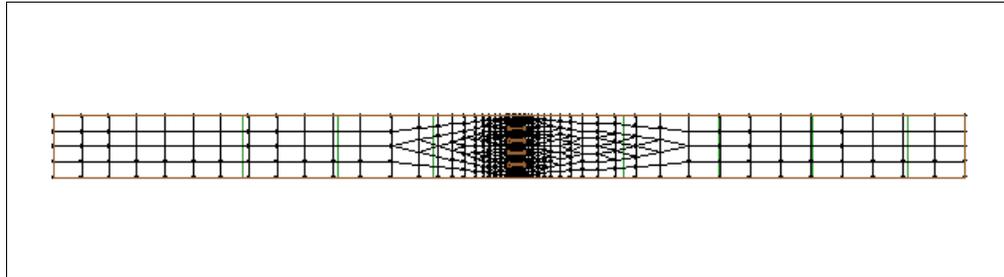


Figure 11.1 The mesh contained in the file *proto.2dm*.

## 11.2 Creating Materials

---

When this mesh was opened, the elements each contained a material type ID. The materials were created with default parameters which must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Reference Manual* for details of what each parameter represents).

There is only one material in this example. To edit the material parameters:

1. Select *HIVEL | Material Properties*.
2. In the *HIVEL Materials Editor*, select the material labeled *Material 1*. Be sure that its ID number is 1 in the top right corner.
3. Enter 0.014 for  $n$  (Manning's roughness).
4. Click the *Close* button.

The material now has the correct parameter associated with it. The material can be displayed by opening the *Display Options* dialog and turning on the *Materials* toggle. If you desire, turn the material on to display it.

## 11.3 Creating Nodestrings

---

Before boundary conditions can be applied at the inflow or outflow boundaries, nodestrings must first be created.

To create the nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. To create the inflow nodestring, click on the node on the top of left side of the mesh. While holding *Shift* key, double-click on the node on the bottom of the left side.

- Repeat step 2 to create the outflow nodestring on the right side of the mesh.

## 11.4 Defining Boundary Conditions

### 11.4.1 General Parameters

The finite element network is only the first part of the numerical model. We have already defined the material properties associated with the different regions of the mesh. These properties help control how water will flow, and thereby allow us to build a model which matches the physical situation. In addition to the geometry and the material properties, we must define several other boundary conditions. To edit the HIVEL boundary conditions:

- Select *HIVEL / Model Control*.
- Enter the necessary values to make all global parameters match those in Figure 11.2. (Select English units first because changing units will automatically update system variables such as gravity).

**HIVEL Model Control**

Job Titles:

Title 1: Files created for the HIVEL2D 2.01 Examples

Title 2: Created by SMS

Title 3: Bridge Piers in High Velocity Channel

Turbulence Coefficients: Normal (smooth): 0.1000 Near jump (rough): 0.5000

Units:  English units  Metric units

Iterations: Max # of iter./timestep: 6 Froude # convergence tol: 0.0050

Specify Petrov Galerkin coeffs.

Upstream weighting of the test function in:

Smooth regions: 0.2500

Rough regions: 0.5000

Computation Time: Time step size: 4.0000 # time steps: 100 Save output every 100 th timestep

Gravitational acceleration (g): 32.1890

Conversion coeff. for Manning's n (Co<sup>2</sup>): 2.2080

Temporal Derivative:  First order backward  Second order backward

OK Reset Defaults Cancel

Figure 11.2 The HIVEL Model Control dialog.

### 11.4.2 Defining Steady State Flow and Head

---

For this tutorial, we will define boundary conditions at the nodelist of the inflow boundary, as well as along a nodelist at the outflow boundary.

To assign the inflow boundary condition:

1. Choose the *Select Nodestrings*  tool from the *Toolbox* and select the nodelist across the inflow boundary on the left side of the mesh
2. Select the *HIVEL / Assign BC*.
3. Select the *Supercritical* and *Inflow string* options.
4. Select *unit discharge, x and y components*, and *depth* for the *Inflow Parameters*.
5. Enter 148.5 for *P*, 0.0 for *Q*, and 7.22 for *depth*.
6. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

To assign the outflow boundary condition on the right side of the mesh:

1. Choose the *Select Nodestring*  tool from the *Toolbox* and select the nodelist across the outflow boundary.
2. Select the *HIVEL / Assign BC*.
3. Select the *Supercritical* and *Outflow string* options.
4. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

Generally for *HIVEL2D*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries. See the *HIVEL2D Reference Manual* for more about assigning boundary conditions.

### 11.4.3 Creating the Hotstart File

---

*HIVEL2D* must have an initial hotstart file to run a solution. *SMS* allows you to create this hotstart file either using constant values or a data set that has been previously loaded through the *Data Browser*. For this tutorial, a data set will be created using the *Data Calculator*.

1. Select *Data | Data Calculator*.

2. Enter “Start Depth” in the *Result* field.
3. Enter 7.22 in the *Expression* field. This is the constant depth.
4. Click the *Compute* button to add “Start Depth” to the Data Sets.
5. Click the *Done* button to leave the *Data Calculator* dialog.

Now the *HIVEL* hotstart file can be created.

1. Select *HIVEL | Build Hot Start*.
2. Enter 0.0 for the Time associated with step m.
3. For both Step m-1 and Step m, select the Constant option for the discharge. Enter a value of 148.5 for  $p$  and 0.0 for  $q$ .
4. For both Step m-1 and Step m, select the Data Set option for the Water Depth. Press the *Select* button.
5. From the *Select Data Set* dialog highlight the “Start Depth” and press the *Select* button to exit.
6. Click the *OK* button to close the *Build Hot Start* dialog.

When *HIVEL2D* starts, it will use these values as an initial starting place. It is important to note that at the end of the computations, this file will be over written. Therefore, a backup copy should be created if desired.

## 11.5 Saving The Simulation

---

*HIVEL2D* uses a geometry file, boundary condition file, and hotstart file written by *SMS* to run an analysis. These files are specified in a *superfile* which is also written by *SMS*. You must save data that has been created.

1. Select *HIVEL | Save Simulation*.
2. Make sure all available options are turned on. Enter the filename "proto\_h" in the *Prefix for all files* edit box, then click the *Update* button.
3. Enter the filename “wse1” in the *WS solution* edit box.
4. Enter the filename “vel1” in the *Flow solution* edit box.
5. Click the *OK* button to save the simulation.

The boundary conditions you specified are now saved. Before continuing with the analysis, check the model for completeness. To do this:

1. Select *HIVEL | Model Check*.
2. Click the *Run Check* button.

The *HIVEL2D* model checker will report two possible warnings. The first is that the mesh has not been renumbered during this session. This can be ignored because it was done in a previous session. The second is that duplicate nodes were found. When working with smaller meshes, the node tolerance may be changed to eliminate this warning.

To change the node tolerance:

1. Select *Nodes | Options*.
2. Enter a value of 0.1 for the tolerance.
3. Click the *OK* button to close the *Node Options* dialog.

## 11.6 Using HIVEL2D

---

The analysis program *HIVEL2D* can be launched from inside SMS. To do this:

1. Select *HIVEL | Run Hivel*.
2. If *SMS* shows a message that the *hivel2d* executable is not found, click the *File Browser*  button to find the *HIVEL* executable.
3. Click the *OK* button to run the model.

*HIVEL2D* may take a few minutes to run this solution. When it is finished, the window will go away. The solution files are *wse1.dat* and *vell.dat*. These can be imported through the *Data Browser*.

## 11.7 Conclusion

---

This concludes the *HIVEL* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Quit*. If prompted to confirm, click the *Yes* button.

---

## ***CGWAVE Analysis***

---

### **12.1 Introduction**

This lesson will teach you how to prepare a mesh for analysis and run a solution for *CGWAVE*. First, make sure you are in the correct portion of *SMS*. To do this:

1. In the *Mesh*  module, select *Data / Switch Current Model*.
2. Select the *CGWAVE* option. If it is unavailable, you will need to run *SMS* in *Demo Mode*.
3. Click the *OK* button.

---

### **12.2 Opening The Data**

You will start with the data file *indiana.xyz* which contains a set of points as shown in Figure 12-1. These points contain depth data from which a mesh will be created. To open the data:

1. Select *File / Import*.
2. Choose the *XYZ Data* option and click the *OK* button.

3. Highlight the file *indiana.xyz* in the *tutorial* directory and click the *Open* button. If a mesh is already open, you will be warned that the previous mesh will be deleted. If this happens, click the *OK* button at the prompt.

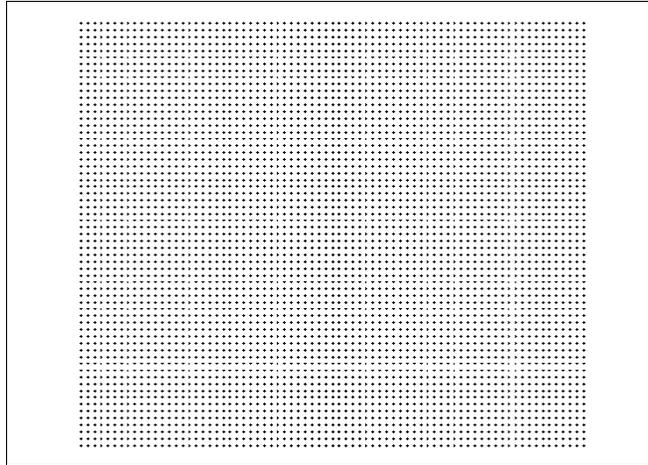


Figure 12-1 The *indiana.xyz* data.

## 12.3 Creating a Wavelength Function

---

The first step in creating a mesh for *CGWAVE* is to create a wavelength function. The wavelength function is an intermediate step to creating a size function, which is explained in section 12.4. The *z* value of each point in the *indiana.xyz* data is actually a water depth value. The wavelength at each point is calculated from this depth value using a complicated equation. It is sufficient to say that a larger wavelength is calculated from a larger water depth value. To create the wavelength function:

1. Select *CGWAVE / Create Functions*.
2. Turn off everything but the *Transition Wavelength/Celerity* option.
3. Leave the function name as *Transition* and enter a *Period* of 20.
4. Click the *OK* button.

Two new data sets will be created, one named *Transition\_Wavelength* and the other named *Transition\_Celerity*. These can be seen in the *Data Browser* (see the *SMS Reference Manual* for more information on the *Data Browser*).

---

## 12.4 Creating a Size Function

---

The size function is created from the wavelength function. The size function is the function which determines the element size that will be created by SMS. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

The wavelength function that was created in section 12.3 contains values that are twice as large as the desired size values. The wavelength function will be scaled by one half to create the size function. To do this:

1. Select *Data / Data Calculator*.
2. In the top left section of the *Data Calculator*, highlight the function named *Transition\_Wavelength* and click the *Add to Expression* button. The letter that represents this function will appear in the *Expression* field.
3. Click the / (divide symbol) in the middle section of the *Data Calculator*.
4. After the divide symbol, enter the number 2 (two) using the keyboard.
5. In the *Result* field, enter the name *size* and then click the *Compute* button.
6. Click the *Done* button to exit the *Data Calculator*.

A new data set named *size* is created which is half the *Transition\_Wavelength* data set. This can be seen in the *Data Browser* (see the *SMS Reference Manual* for more information on the *Data Browser*).

---

## 12.5 Creating a Background Scatter Set

---

The xyz data, along with all of the data sets that have been created, needs to be converted to a *scattered data set* so that it can be used to determine element sizes. Scattered data sets are used as background information from which a mesh is generated. To convert the xyz points to a scattered data set:

1. Select *Data / Mesh->Scatterpoint*.
2. Click the *OK* button in the prompt with the default name of scatter.

The screen will refresh with one scatter point on top of each xyz point. The scattered data set contains a copy of all functions that were created in sections 12.3 and 12.4 as well as a copy of the original water depth function.

## 12.6 Defining The Domain

---

A domain represents the region that is offshore. In *CGWAVE*, the domain can be a circular, semi-circular, or rectangular region. In *SMS* a *Feature Arc* is used to define the coastline. After the coastline is defined, *Feature Arcs* and *Feature Polygons* are used to define the domain region.

### 12.6.1 Creating the Coastline

---

*SMS* can automatically create a coastline at a specific elevation or water depth from a scattered data set. The active function of the active scattered data set will be used for this operation. You should currently have only one scattered data set. To make the elevation function active:

1. Switch to the *Scatterpoint*  module.
2. Select *Data / Data Browser* and highlight the function named *elevation* in the top window of the *Data Browser*. Remember that this function was converted with the xyz data.
3. Click the *Done* button to exit the *Data Browser*.

Before creating the coastline, you should create a *CGWAVE* coverage. To do this:

1. Switch to the *Map*  module.
2. Select *Feature Objects / Coverages*.
3. Change the *Coverage type* to *CGWAVE*, and click the *OK* button.

With the coverage type set and the active scatterpoint data set defined, you are ready to create the coastline. To create the coastline arc:

1. Select *Feature Objects / Create Coastline*.
2. You will be prompted for an elevation value. Enter the value of 1.0 and click the *OK* button to this prompt.

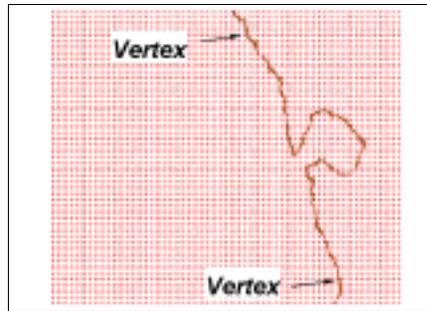
After a few seconds, the display will refresh with an arc representing the 1.0 water depth line, as shown in Figure 12-2a.

### 12.6.2 Creating the domain

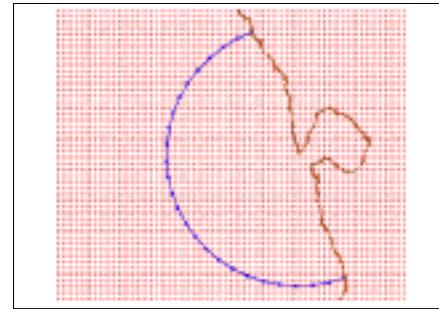
---

With the coastline created, you are ready to create the mesh domain. This model will use a semi-circular domain which intersects with the coastline. To create the domain:

1. Choose the *Select Feature Verticies*  tool from the *Toolbox*.
2. Hold the *SHIFT* key and click on two vertices, one at each end of the coastline arc, as shown in Figure 12-2a.
3. Select *Feature Objects / Define Domain*. Select *Semi-circular*, and click *OK*. This creates a semicircular *Ocean* arc as shown in Figure 12-2b.



(a). The coastline feature arc.



(b). The domain feature arc.

Figure 12-2 The indiana scatterpoint and feature data.

Now that feature arcs define the domain, a feature polygon must be created from the feature arcs. To create the polygon:

- Select *Feature Objects / Build Polygons*. You need to build polygons from all available arcs, so click the *OK* button at the prompt.

After this command is executed, polygons are formed from any set of arcs which form a closed loop. The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created. For this example, there should now be a single polygon made from the semi-circular ocean arc and the part of the coastline arc which it intersects.

## 12.7 Creating The Finite Element Mesh

There are various automatic mesh generation techniques which can be used to create elements inside a specified boundary. One of these is applied to each polygon, after which a finite element mesh can be generated. For this tutorial, there is only one polygon, which will be assigned the *Density mesh* type.

### 12.7.1 Setting Up The Polygon

When using density meshing, *SMS* determines element sizes from a *size function* in a scattered data set. The size function to be used in this example was created back in section 12.4. To set up the feature polygon for density meshing:

1. Choose the *Select Feature Polygons*  tool from the *Toolbox*. With this tool selected, double-click inside the polygon which defines the domain.
2. In the *Polygon Attributes* dialog, change the *Mesh Type* to *Density* and press the *Scatter Opts* button.
3. In the top right of the *Scatter Options* dialog, highlight the function named *size*. This sets the size function as that created in section 12.4.
4. In the bottom left of the *Scatter Options* dialog, turn on the *Truncate values* option and set the *Min* and *Max* to 10 and 1000. This sets up a minimum and maximum size to be used when creating elements.
5. Click the *OK* button to close the *Scatter Options* dialog.
6. In the *Polygon Attributes* dialog, click the *Elevation Scatter Opts* button.
7. In the *Scatter Options* dialog, highlight the function named *elevation*. as the scatter function and make sure the *Truncate Values* option is turned off. As mesh nodes are created, their elevation value will be assigned from the original water depth values that were read from the xyz file in section 12.2.
8. Click the *OK* button to close both dialogs.

The polygon is now set up to generate finite elements inside the boundary. When more than one polygon exists, the meshing attributes need to be set up for each of the polygons.

### 12.7.2 Generating The Elements

---

Since there is only one polygon in this example, you are ready to have *SMS* generate the finite element mesh from the defined domain. To create the mesh:

1. Select *Feature Objects / Map->2D Mesh*.
2. Make sure the *Mesh Source*, *Bathymetry Source*, *Delete Existing Mesh*, and *Display Meshing Process* options are all on.
3. Under *Mesh Source*, the *Mesh From Poly* option should be selected and under *Bathymetry Source*, the *Scatter Point* option should be selected.
4. Click the *OK* button to start the mesh creation.

With the *Display Meshing Process* option turned on, you will see elements being created on the screen. The element size will become smaller as the mesh gets closer to the coastline boundary due to the defined size function. When the meshing process is complete, the finite element mesh will look like Figure 12-3.

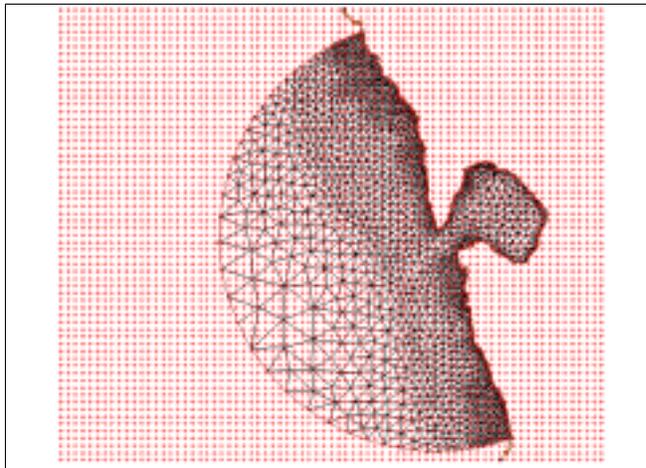


Figure 12-3 The completed finite element mesh.

At this point, the display is quite cluttered with all the data that has been created. By changing some display settings, the display can become less cluttered. To change the display settings:

1. Switch to the *Scatterpoint*  module and select *Display / Display Options*. In the *Scatterpoint Display Options* dialog, turn off the *Scatterpoint Symbols* option. Click the *OK* button.
2. Switch to the *Map*  module and select *Feature Objects / Display Options*. Change the status of the coverage to be hidden. Click the *OK* button.
3. Switch to the *Mesh*  module and select *Display / Display Options*. Turn off everything except the *Elements* and *Contours* options.
4. Click the *Options* button which is next to the *Contours* option. Change the *Number of Intervals* to 20. Change the *Contour Method* to *Color fill between contours*.
5. Click the *OK* button in both dialogs.

The display will refresh after steps 1, 2, and 5 above. After the final refresh of the display, you will see contours of water depth with the elements drawn on top of those. You can clearly see that as the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.

## 12.8 Model Control

When creating a *CGWAVE* model, the boundary conditions are wave amplitude, phase, and direction. To define these *incident wave conditions*:

1. Select *CGWAVE / Model Control*.
2. Click the *Import* button under the *Incident Wave Conditions* and select the file *indianawav.dat*. A set of previously created wave conditions are opened and displayed in the *Incident Wave Conditions* window.
3. Click the *OK* button to exit the *CGWAVE Model Control* dialog.

## 12.9 Saving the CGWAVE Data

---

*CGWAVE* uses a geometry file and a 1-D file to run an analysis. This file consists of two lines that run perpendicular from the coastline to the extents of the domain. The 1-D file is generated automatically by *SMS* using the active scatter set. The file contains depth information on both sides of the domain. To save these files:

- Select *CGWAVE / Save Simulation* and enter the name *indianaout.dat*.

## 12.10 Running CGWAVE

---

*CGWAVE* can be run from *SMS*. To run *CGWAVE*:

4. Select *CGWAVE / Run CGWAVE*.
5. Click on the file button and find the *CGWAVE* executable. Click the *OK* button.
6. Respond to the prompts that appear as follows:

```
PLEASE SPECIFY INPUT DATA FILENAME :
indianaout.dat
PLEASE SPECIFY OUTPUT WAVE POTENTIAL FILENAME :
indiana.sol
A PREVIOUS SOLUTION FILE AS AN INITIAL STATE (Y/N)?
n
```

## 12.11 Conclusion

---

This concludes the *CGWAVE* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.

---

## ***ADCIRC Analysis***

---

### **13.1 Introduction**

This lesson will teach you how to prepare a mesh for analysis and run a solution for *ADCIRC*. You will be using the files *ponce.xyd* and *ponce.cst*. The first file contains a set of scattered data points that will be used as background data for creating the finite element mesh. To open this file:

1. Select *File | Open*.
2. Open the file *ponce.xyd* from the *tutorial* directory.

---

### **13.2 Creating the Mesh**

As with *CGWAVE*, *ADCIRC* works very closely with the map module. Before continuing, a *CGWAVE* coverage needs to be created. To do this:

1. Switch to the *Map*  module.
2. Select *Feature Objects | Coverages*.
3. Change the *Coverage type* to *ADCIRC* and click the *OK* button.

In lesson 12, a coastline was automatically created by *SMS*. This example will show how to open a coastline file. The coastline file is called *ponce.cst*. To open the file:

1. Select *File / Import*.
2. Choose the *Coastline* option and click the *OK* button.
3. Open the file *ponce.cst*.

After the file is opened, the display will refresh, showing a feature arc which represents the coastline of the model, as shown in Figure 13-1. The arc was automatically defined as a *Mainland* arc, meaning it is a coastline.

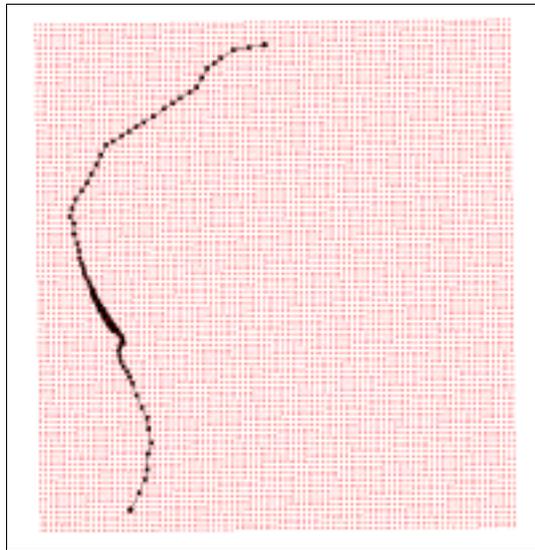


Figure 13-1 The coastline arc from *ponce.cst*.

### 13.2.1 Creating The Domain

---

The domain should be defined by a closed loop of arcs which form a polygon. For this example, the domain is located to the right of the coastline. The domain does not need to be any specific shape, so you can simply create arcs out in the domain area. To create the domain:

1. Choose the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create an arc starting on one end of the coastline, extending out into the ocean, and then connecting to the other end of the coastline, as shown in Figure 13-2.
3. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
4. Double click on the new arc and change the *Boundary Type* to *Ocean*, then click the *OK* button.

5. Click on an empty part of the screen to deselect all arcs, and select *Feature Objects / Build Polygons*.

The domain is now ready for a mesh to be created.

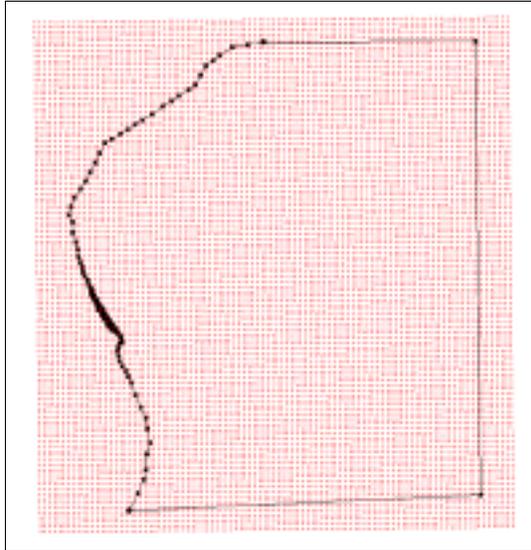


Figure 13-2. The domain for the model.

### 13.2.2 Defining Mesh Generation Parameters

A finite element mesh will be generated inside the polygon according to the parameters that are assigned to it. There are various automatic mesh generation algorithms that can be applied to feature polygons. See the *SMS Reference Manual* for more information on these algorithms.

A good mesh generation algorithm for coastal models is called *Density Meshing*, which determines the element size from a function in an underlying scattered data set. The function used with density meshing is often the bathymetry function. This is advantageous because the bathymetry is small at the coast and gets deeper out in the ocean. Using this as the size function will create smaller elements at the coast and larger elements in the ocean. To assign the meshing parameters:

1. Choose the *Select Feature Polygon*  tool.
2. Double-click inside the polygon to open the *Polygon Attributes* dialog.
3. Change the *Mesh type* to *Density*.
4. Click the *Scatter Opts* button and set the *Default extrapolation* value to 10.0.
5. Turn on the *Truncate values* option with a *Min* of 3 and *Max* of 1000.

6. Click the *OK* button to close both dialogs.

### 13.2.3 Generating The Finite Element Mesh

---

After setting up the automatic mesh generation parameters, *SMS* can generate the finite element mesh inside the polygon. To do this:

1. Select *Feature Objects / Map -> 2D Mesh*.
2. Make sure the *Mesh source* option is on and set to *Mesh from poly* so that the mesh will be created using the options just set.
3. Make sure the *Bathymetry source* option is on and set to *Scatter point* so that the nodal depth values will come from the scattered data set.
4. If you want, turn on the *Display meshing process* option. It is interesting to see the meshing process, but it also slows the process down.
5. Click the *OK* button to generate the mesh.

It will take a few minutes to generate the finite element mesh. When *SMS* is done, the generated mesh should look like Figure 13-3. The boundaries of the mesh match the polygon boundaries. During this process, two nodestrings were created, one on the *Ocean* boundary and one on the coast, or *Mainland*, boundary.

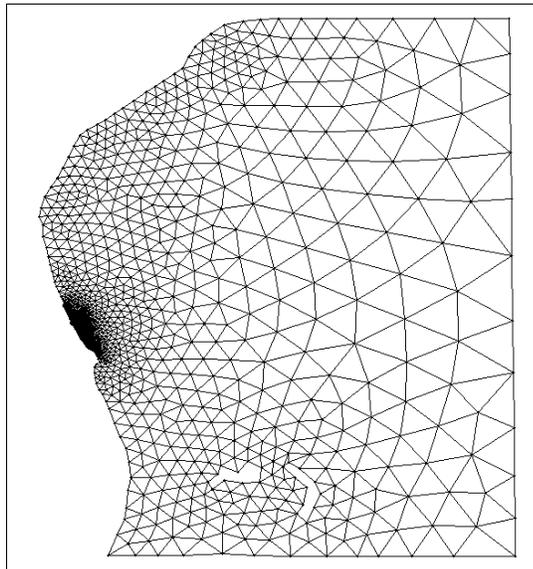


Figure 13-3. Finished mesh.

### 13.3 Setting Model Parameters

---

Model parameters are set in the *ADCIRC Model Control* dialog. For speed purposes, the model controls for this simulation will be read from a file. This will also assure that the finite element mesh you use is the same as that in the tutorial. A new finite element mesh and matching model control file need to be opened. To do this:

1. Switch to the *Mesh*  module.
2. Select *ADCIRC / Open Simulation*.
3. Turn on the *Open grid file* option and click the *File Browser*  button.
4. Open the file named *ponce.grd* from the *tutorial* directory.
5. Similarly, open the *Open control file* named *poncectl*.
6. Click the *OK* button to open the files.

The geometry should look like the one previously shown in Figure 13-3. The model parameters that were opened can be viewed in the *ADCIRC Model Control* dialog. To view these parameters:

- Select *ADCIRC / Model Control*.

After viewing the various options, click the *OK* button.

### 13.4 Saving The Simulation

---

*ADCIRC* uses the mesh and the model control files to run an analysis. To save the simulation files:

1. Select *ADCIRC / Save Simulation*.
2. Enter the name *ponceout* in the *Prefix* field and click the *Update* button.
3. Click the *OK* button to save the simulation.

### 13.5 Running ADCIRC

---

*ADCIRC* is not public domain, therefore if you have not registered for it, you will not have the program on your disk. If you have registered to use *ADCIRC* and you are not running *SMS* in *Demo* mode, *ADCIRC* can be launched from *SMS*. However, the mesh and model control files must currently be given specific names for the

program to run. The file with the finite element network, or grid file, must be named *fort.14*, while the model control file must be named *fort.15*. To run *ADCIRC*:

1. Copy the file *ponceout.grd* to the name *fort.14*.
2. Copy the file *ponceout.ctl* to the name *fort.15*.
3. Select *ADCIRC / Run ADCIRC*.
4. If a message is given that the *ADCIRC* executable is not found, click the *File Browser*  button and manually find the executable file.
5. Click the *OK* button to launch the *ADCIRC* analysis.

This model can take many hours to run on a Pentium II 300 MHz computer. Even if you have registered the *ADCIRC* program, you may want to skip ahead with the solution file that is supplied with *SMS*.

## 13.6 Viewing Solution Files

---

Post-processing is discussed in lesson 6. However, there are four files that *ADCIRC* creates which will be discussed here. These files are *fort.63*, *fort.64*, *fort.53*, and *fort.54*. The first two files can be imported through the *Data Browser* using the *ADCIRC file* option, while the latter two files can be imported using the *ADCIRC harmonic file* option. After these simulations are imported to *SMS*, they can be manipulated and viewed just as other data set files.

## 13.7 Conclusion

---

This concludes the *ADCIRC* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.

---

## ***Basic WSPRO Analysis***

---

### **14.1 Introduction**

---

*WSPRO* is a one-dimensional water-surface profile computation model that can be used to analyze gradually varied, steady flow in open channels. *WSPRO* can be used to analyze flow through bridges and culverts, embankment overflow, and multiple-opening stream crossings.

This tutorial demonstrates the one-dimensional section editing capabilities of *SMS* and the process involved in performing a *WSPRO* analysis and viewing the solution inside *SMS*.

---

### **14.2 Setting Up SMS For WSPRO**

---

The *WSPRO* interface is located in the *River* module of *SMS*. This module uses both the *River Window* and the *Plot Window*. To set up the *SMS* interface for working with *WSPRO*:

1. Switch to the *River*  module.
2. Select *Display / River Window*.
3. Select *Display / Plot Window*.

## 14.3 Defining a Cross Section

---

As a one-dimensional model, *WSPRO* requires parameters to be lumped together as location data. These locations are referred to as *sections*. A new section needs to be created at a proposed bridge site.

### 14.3.1 Creating The Section

---

Before importing a set of points to define the cross section, the section must be created. To create a new *WSPRO* section:

1. Choose the *Create River Sections*  tool from the *Toolbox*.
2. Click inside the *River Window*.
3. In the prompt that opens, enter a *Section reference distance* (SRD) value of 2485. Click the *OK* button in this prompt.

The SRD value is the distance upstream from an arbitrary datum. All SRD values must be positive and must increase as you go upstream. When the section is created, an icon representing the section appears in the *River Window* and a trapezoidal cross section is shown in the *River Cross Section* plot of the *Plot Window*.

### 14.3.2 Importing Geometry Points

---

The section points need to be redefined to the correct shape of the cross section. A ground survey was performed at the site location. To import the survey data:

1. Select *WSPRO / List Geometry Points*.
2. Click the *Import* button and open the file *xsec.dat* from the *tutorial* directory. This file contains a list of location vs. elevation points.
3. Click the *OK* button to update the plot, as shown in Figure 14-1.

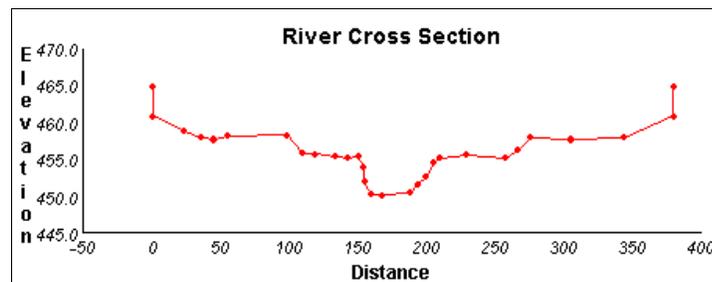


Figure 14-1 The *xsec* cross section data.

### 14.3.3 Assigning Section Parameters

After defining the cross section geometry, other attributes can be assigned to the section. For this section, a name and valley slope should be defined. To do this:

1. Select *WSPRO / Section Attributes*.
2. Enter the name “FULLV” for the section, signifying it is a *bridge full valley* section. *WSPRO* section names cannot be longer than five characters.
3. Turn on the *Valley slope* option and enter a value of 0.0012. This defines the bed slope upstream of the cross section.
4. Click the *OK* button to close the *Cross Section* dialog.

## 14.4 Interpolating Cross Sections

Before defining a bridge at the full valley section, *WSPRO* requires an *approach* section and an *exit* section to be defined. The river cross section does not change significantly in such a short distance, so the survey data of the full valley section will be valid for the approach and exit sections, with a slight elevation change for the bed slope. The approach should be located one bridge length upstream from the bridge and the exit should be located one bridge length downstream from it.

Since we do not yet know the bridge dimensions (we are designing it according to the *WSPRO* simulation results), an initial length of 120 ft and width of 30 ft will be used as a guess. The approach section must therefore be 120 ft from the upstream face of the bridge and the exit section must be 120 ft from the downstream face. The *SRD* value at the bridge section corresponds with the downstream bridge face, so the approach *SRD* value must be 120 larger than the full valley *SRD* and the exit *SRD* value must be 120 smaller than the full valley *SRD*, as shown in Figure 14-2.

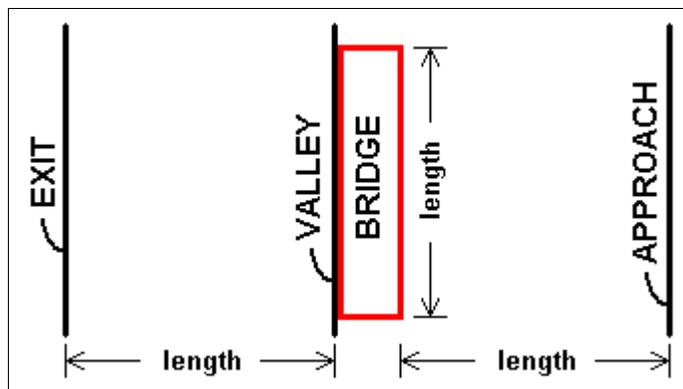


Figure 14-2 The *SRD* locations for the three sections.

When creating a new section, *SMS* interpolates/extrapolates the cross sectional data from the existing section(s). To create the approach section:

1. The *Create Sections*  tool should still be selected in the *Toolbox*.
2. Click in the *River Window* ABOVE the *FULLV* section icon.
3. In the prompt, enter a value of 2635 feet (2485+30+120) for the SRD value and click the *OK* button.

*SMS* will create a new section whose cross sectional geometry is extrapolated from that of the full valley section, using the new SRD value and the valley slope. The *Plot Window* will update to show the new section. To create the exit section:

1. Click in the *River Window* BELOW the *FULLV* section icon.
2. Enter an SRD value of 2365 (2485 – 120) and click the *OK* button.

Once again, a new section will be created, this time below the full valley section location. The *Plot Window* will update to show the new section. The two new sections have the same valley slope that was assigned to the full valley section, but the new sections need better names. To rename the sections:

1. Choose the *Select Section Points*  tool from the *Toolbox*.
2. In the *River Window*, double-click on the section above the full valley section. Give it the name of “APPR” and click the *OK* button.
3. Double-click on the section below the full valley section. Give it the name of “EXIT” and click the *OK* button.

## 14.5 Defining Material Properties

---

The cross sectional geometry has just been defined at the proposed bridge location. Manning’s roughness values for each cross section must now be assigned. These values can be applied to the downstream section and then WSPRO will propagate them upstream to the other sections. As with the cross section shape, the roughness values of each section are assumed to be identical.

### 14.5.1 Creating Section Break Points

---

Section break points (*SA points*) are used to define points in the cross section where changes in roughness occur. To create section break points at the exit section:

1. Choose the *Create Section Break*  tool from the *Toolbox*.

2. Select the *EXIT* section by clicking on it in the *River Window*.
3. Add four *SA* points by clicking near the following locations in the *Plot Window*: 99, 150, 210 and 276. (The cursor location is shown in the *Edit Window* as the mouse is moved.)

When creating *SA* points, it is virtually impossible to click in the exact location, but *SMS* allows you to move an *SA* point after it is created. To move the *SA* points:

1. Choose the *Select Section Breaks*  tool from the *Toolbox*.
2. Click on the first section break that should be at location 99.0. The exact break location is shown in the *Edit Window*.
3. Change the *SA* point location to 99.0. Reset all four *SA* points to their correct locations shown above.

With all *SA* points in the proper locations, the display should look like Figure 14-3.

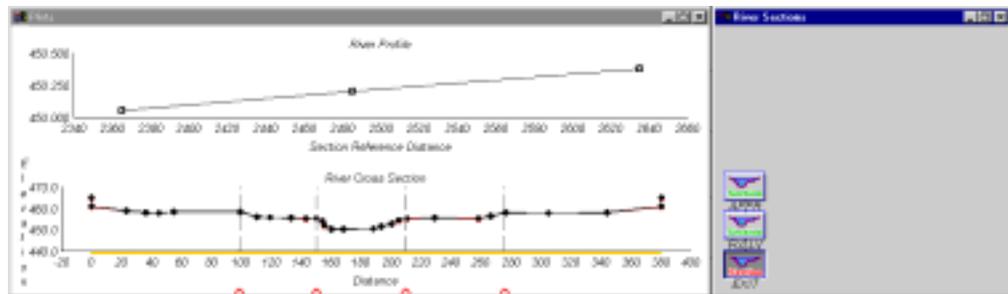


Figure 14-3 The sections with *SA* points created.

## 14.5.2 Creating Materials

A material type is assigned to each part of the cross section between *SA* points. The material type defines the roughness values to be used for each part of the cross section. *SMS* starts with a *Disable* material and with one other material defined. For this example, four materials are needed. To create three more materials:

1. Select *Edit / Materials Data*. You should see two materials in the window, one named *Disable* and one named *material 01*.
2. Click the *New* button three times to create three new materials. You should now see five materials: *Disable*, and materials one through four.
3. Click the *Close* button to close the *Materials Data* dialog.

### 14.5.3 Defining Material Roughness

Roughness values can now be assigned to each of the materials. In *WSPRO*, either a single roughness value is assigned or two roughness values with corresponding depths (depth breakpoints) are assigned. The figure in the *WSPRO Material Editor* dialog shows how these values are used. To define the roughness values:

1. Select *WSPRO / Roughness Parameters*.
2. In the *WSPRO Material Editor* dialog, set the values as shown in Table 14-1. (Turn on the *Use depth breakpoints* for materials 1, 2, and 4.)
3. Click the *Close* button to close the *WSPRO Material Editor* dialog.

Table 14-1 Roughness properties for the material types.

| Material | N1    | D1  | N2   | D2  |
|----------|-------|-----|------|-----|
| 1        | 0.055 | 1.0 | 0.05 | 3.0 |
| 2        | 0.065 | 2.0 | 0.06 | 5.0 |
| 3        | .04   | -   | -    | -   |
| 4        | 0.065 | 1.0 | 0.06 | 4.0 |

### 14.5.4 Assigning Materials To a Section

The materials that have just been created can now be assigned to different parts of the *EXIT* cross section. To assign the materials:

1. Choose the *Select Material Regions*  tool from the *Toolbox*.
2. Click between the left cross section edge and the first *SA* point.
3. In the *WSPRO Material Editor*, highlight the material named *material 01* and click the *Close* button.
4. Repeat this process, highlighting the material as shown in Figure 14-4.

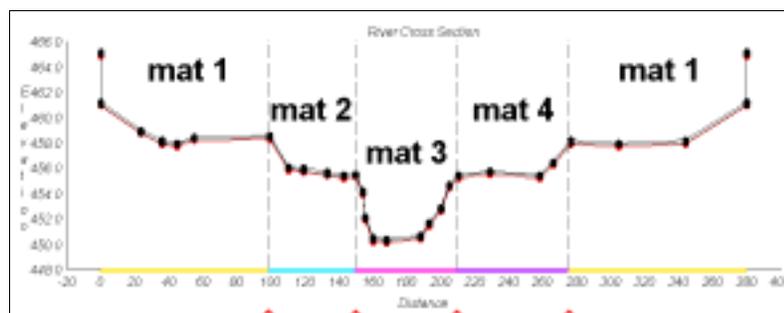


Figure 14-4 The materials to assign.

A color bar at the bottom of the *Plot Window* indicates the material that is assigned to each section. These colors can be changed through the *Materials Data* dialog.

## 14.6 Model Parameters

---

*WSPRO* can compute multiple profiles in a single simulation. Each profile requires at least one flow, a downstream water surface or a friction slope, and a guess that the flow will be subcritical or supercritical. Title information can also be defined. For this example, only one profile will be computed. To define the profile:

1. Select *WSPRO / Run Control*.
2. Enter titles if desired to describe this simulation. These should be something like “Sample WSPRO Problem” or “Simple Bridge Opening”.
3. If the *EXIT* section does not appear in the left *Flow Sets* area, highlight the section and click the  button. In a *WSPRO* profile, the flow rate must be specified at the most downstream section. It may also be specified at other sections, such as when an inlet enters the river. However, there must always be a downstream flow rate specification.
4. Click the *Add* button to create a new profile.
5. Assign a  $Q$  (flow) of 5500 cfs, an initial *Friction slope* of 0.002 ft/ft, and set the flow type as *Subcritical*.
6. Click the *OK* button to close the *WSPRO Run Control* dialog.

## 14.7 Saving a Simulation

---

All the required data has now been entered for an initial simulation of the river reach. To save the simulation:

1. Select *WSPRO / Save Simulation*.
2. Save the simulation as *basic.dat*.

## 14.8 Running The WSPRO Simulation

---

The simulation file that was saved contains the sectional properties with roughness values and global parameters. To run *WSPRO* on the simulation:

1. Select *WSPRO / Run WSPRO*.
2. *SMS* finds and displays the execute path for the *WSPRO* program. If it cannot be found, or if *SMS* finds a version that you don't want to use, click the *File Browser*  button to find and select the proper executable file.
3. Click the *OK* button to run *WSPRO*.

*SMS* will launch *WSPRO* in a new window. The window will go away after a few seconds when execution is complete.

## 14.9 Viewing The Profile

---

*WSPRO* creates two output files. The *.plt* file contains the solution results. These results can be viewed inside *SMS* by importing the file through the *Data Browser*. The *.lst* file is a text file that stores tables of solution variables. It can be viewed in any text editor.

### 14.9.1 Viewing The .LST File

---

You should always look at the *.lst* file generated by *WSPRO* to make sure the run was complete. To do this:

1. Select *Display / View Data File*.
2. Open the file *basic.lst* from the file browser.
3. Enter the name of a text editor to use. The default is *notepad* for the *PC* and *vi* for *UNIX* platforms. Click the *OK* button to view the file.

The *.lst* file contains an echo of the input data, the standard profile solution data, and any user-specified tables (there were none in this example). At the end of the file, solution data should be output for each of the three cross sections. Each section is labeled to show variables such as *WSEL* (water surface elevation). The last line in the file should read:

```
*** Normal end of WSPRO execution ***  
*** Time required to run the model ***
```

### 14.9.2 Viewing The .PLT File

---

After assuring that the run was successful, the *.plt* file can be imported into *SMS* and plotted in the *Plot Window*. To import the solution:

1. Select *Data / Data Browser*.

2. In the *River Data Browser*, click the *Import* button and open the file *basic.plt*. The profile will be listed at the top left of the dialog.
3. Turn on the *Visible* option to make the profile visible. The available functions appear in the lower left window of the dialog.
4. By default, only the *geometry* function is visible. Turn it off and turn on the *wsel*, *cwsel*, and *egl* functions.
5. Click the *Done* button to exit the *River Data Browser* dialog.

Three plots will appear in the *River Profile* plot of the *Plot Window*, as shown in Figure 14-5. These plots show the predicted water surface elevation for each of the sections. The predicted values are reported by *SMS* in the *Edit Window*. As the cursor is moved over the plot, the coordinates are updated.

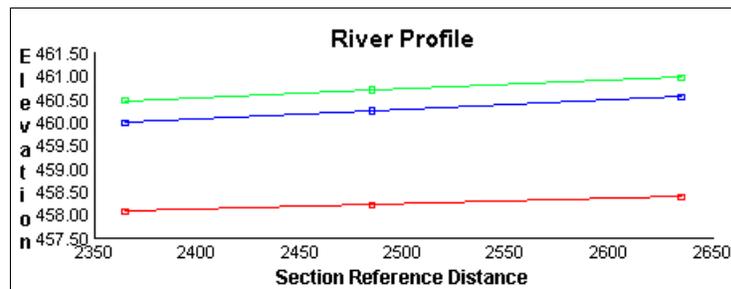


Figure 14-5 The *basic.plt* profile plots.

The color of the curve used to draw each function can be changed using the *WSPRO Profile Options* dialog. To change the function colors:

1. Select *Display / Display Options*.
2. In the *WSPRO Profile Options* dialog, highlight the function named *wsel*. The four boxes at the bottom left will show the attributes for this function.
3. Click the *Color* box and choose a new blue color. Then click the *OK* button. The new color is assigned to *wsel* function.

Other attributes, such as line thickness and symbol type, can similarly be changed.

## 14.10 Adding a Bridge

A bridge will now be added at the original cross section. To do this:

1. Choose the *Create Bridges*  tool from the *Toolbox*.

2. Click on the cross section named *FULLV*. This creates a bridge at the full valley section whose elevation is the average elevation of the cross section and whose length is one-fourth the cross section length.
3. Select *WSPRO / Section Attributes* to edit the bridge attributes. In the *Bridge Section* dialog, set the *Abutments* and *Embankment* to *Vertical* and specify a *Deck width* of 30 feet.
4. Click the *Parameters* button. In the *Bridge Component Mode Parameters* dialog, assign a *Length* of 120 feet and constrain the abutment locations to 135 feet (*XCONLT*) and 225 feet (*XCONRT*). Set the *Elevation of bottom of deck* to 462 feet.
5. Click the *OK* button in both dialogs to accept the bridge.

The *Plot Window* will update with the new bridge drawn at the full valley section, as shown in Figure 14-6. The bridge length and deck elevation are two variables that can be altered to help satisfy regulatory agency specifications when the solution is attained.

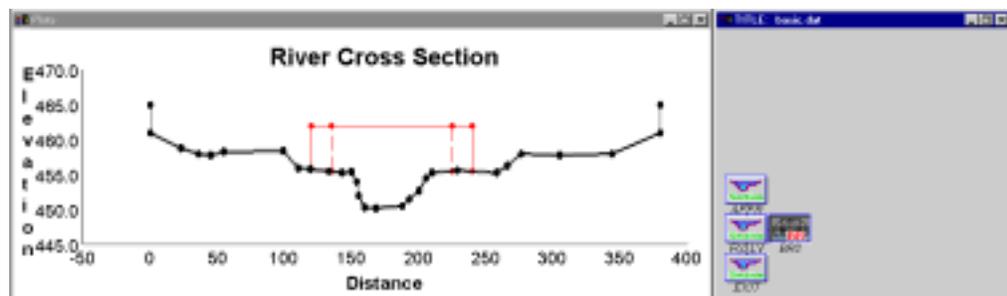


Figure 14-6 The bridge section has been built.

After entering the bridge parameters:

1. Save the new simulation as *wp\_br1.dat*.
2. Run *WSPRO* on the simulation (see section 14.8).

With the addition of the bridge, *WSPRO* now computes two sets of results. The first set is called the contracted, or constrained, solution. This takes into account any backwater generated by the flow going through the bridge. The second set of results is the uncontracted, or unconstrained, solution. This would be the solution if the bridge were not there. It is equivalent to what was computed in section 14.8.

## 14.11 Checking Backwater And Deck Clearance

A bridge has been created with an initial guess for the length and height. We want to minimize construction costs by minimizing the length and height of the bridge, but we must also meet the following regulation:

- The maximum allowable backwater generated by the bridge for 5500 cfs is one foot.

The amount of freeboard, or clearance between the water surface and the bridge deck, is often regulated in addition to the backwater. However, this example will use only the backwater regulation. Backwater is measured at the approach section and freeboard is measured at the bridge section, both using the contracted solution.

### 14.11.1 Using Visual Analysis

The design values can be visually evaluated by examining plots of the solution. To view the solution that was created with the bridge:

1. Choose *Data / Data Browser* and import the *wp\_br1.plt* file.
2. Make this new solution visible and hide all functions except for the *wsel*.
3. Turn on the *Show Unconstrained* option for the *wsel* function.
4. Click the *Done* button to update the *Plot Window* as shown in Figure 14-7.

The *Plot Window* will draw two curves, the contracted and unconstrained water surface elevation through the reach. By default, the contracted plot has hollow boxes at data points, while the unconstrained plots have filled boxes at data points. There are two dots at the bridge section on the contracted water surface elevation plot. These represent the upstream and downstream faces of the bridge.

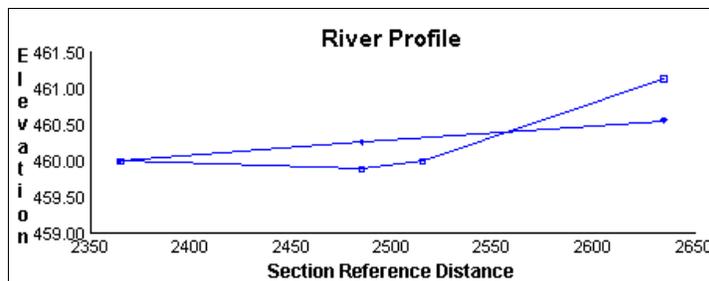


Figure 14-7 The Unconstrained vs. Constrained water surface.

The backwater created by the bridge is determined by the difference in water surface elevation at the approach section. The value of each point can be seen by moving the mouse cursor over them and looking at the *Y coordinate* value reported in the *Edit*

*Window*. The difference can be seen to be about half a foot. Since more backwater can be allowed (up to one foot), the bridge can be shorter.

### 14.11.2 Using the .lst file

---

The visual analysis in the *Plot Window* helps to understand the model results. In addition, the exact values can be seen in the *.lst* file. To see the exact water surface elevation values:

1. Select *Display / View Data File* and open the file *wp\_br1.lst*.
2. Scroll down toward the end of the file to the string “Beginning 1 Profile Calculation(s)”. This is followed by the profile data for the simulation.
3. Find the first block of numbers. This is the unconstrained section of values. The *WSEL* value for the *APPR* section should be 461.408 ft.
4. Find the second block of numbers. This is the constrained section of values. The *WSEL* value for the *APPR* section should be 461.878 ft.
5. The difference in these values is given on a line after this second block of numbers as: *Change in Approach Section Water Surface Elevation: .471*.

This confirms the visual results of about half a foot of backwater generated at the approach section.

### 14.12 Changing Model Parameters

---

Since the backwater generated is less than the one foot limit, the bridge design can be shortened. To change the bridge length:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. In the *River Window*, double-click the bridge.
3. In the *Bridge Section* dialog, click the *Parameters* button.
4. In the *Bridge Component Mode Parameters* dialog, change the *Length* of the bridge to 100.0 ft.
5. Click the *OK* button to close both dialogs.

## 14.13 Finishing The Design

---

With the bridge length changed, do the following:

- Save the file as *wp\_br2.dat* and run *WSPRO*.
- Check the backwater generated
- Redesign the bridge until the backwater generated is approximately 1.0 ft.

The bridge length is only one parameter on the bridge design. There are many other design parameters such as deck elevation, abutment type, and entrance loss type. All of these affect the analysis results. In addition to the bridge, culvert and roadway sections can be defined. Culverts are used to add conveyance, while roadways are used to model overtopping.

## 14.14 Conclusion

---

This concludes the Basic WSPRO Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## ***Advanced WSPRO Analysis***

---

### **15.1 Introduction**

---

*WSPRO* is a one-dimensional water-surface profile computation model that can be used to analyze gradually-varied, steady flow in open channels. *WSPRO* can be used to analyze flow through bridges and culverts, embankment overflow, and multiple-opening stream crossings.

In this lesson a *WSPRO* model will be created by extrapolating cross sections from 2D topography. The model will then be run through the *WSPRO* analysis package.

---

### **15.2 Reading a Tiff Image**

---

A background will be used as a reference, just as was done in lesson 2. To import and register the image:

1. Select *File / Import*.
2. In the *Import Format* dialog, select the *Tiff* option and click the *OK* button.
3. Open the file *redfox.tif* from the *tutorial* directory. The *Register Image* dialog will open, showing the image to be imported.
4. Set the registration point locations to those shown in Table 15-1. Then click the *OK* button.

The tiff image will be resampled and then the display will refresh with the image, as shown in Figure 15-1.

Table 15-1 Image Point Locations

|    | Point 1 | Point 2 | Point3 |
|----|---------|---------|--------|
| U: | 330     | 330     | 2970   |
| V: | 2296    | 254     | 254    |
| X: | 0.0     | 0.0     | 2586.2 |
| Y: | 2000.0  | 0.0     | 0.0    |

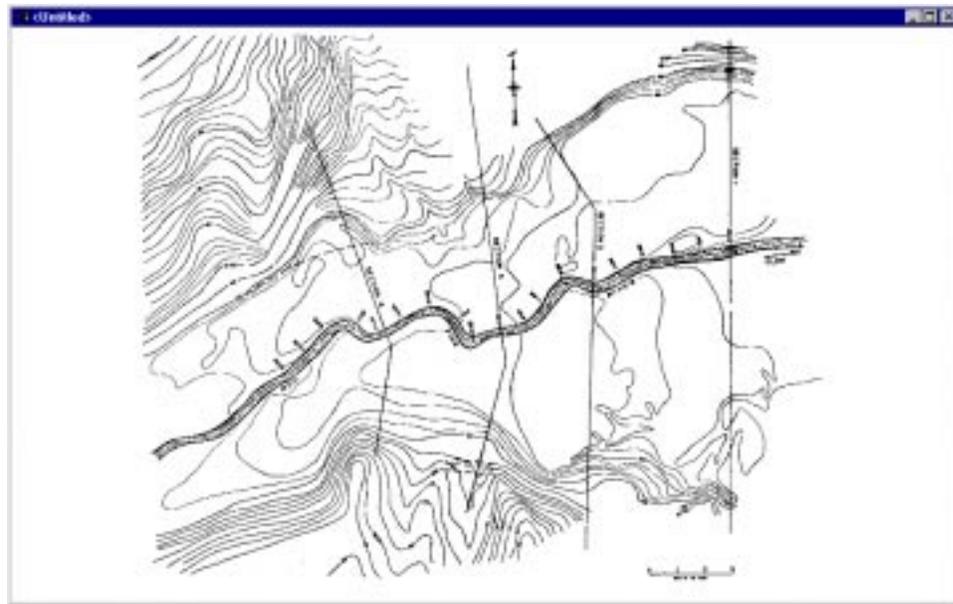


Figure 15-1 The redfox tiff image.

### 15.3 Creating a Conceptual Model

The *Map* module is used to create *conceptual models* of the flow area. The conceptual model uses various *coverages*. Each coverage contains *feature points*, *feature arcs*, and *feature polygons*, which define a set of related data. See the *SMS Reference Manual* for more information on the *Map* module. The first coverage that will be created is for the river sections. To create this coverage:

1. Switch to the *Map*  Module.

2. Select *Feature Objects / Coverages*. Change the *Coverage type* to *WSPRO*. The *WSPRO* coverage is used to create feature objects related specifically to one-dimensional models.
3. Each coverage can be given its own unique name. Enter the name *One D* for this coverage.

### 15.3.1 Defining a Centerline

A centerline arc is used to define the river profile. The first feature arc that is created in a *WSPRO* coverage is always assigned as a centerline. To create this arc:

1. Choose the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create the centerline arc by clicking out a line in the middle of the river. Start the arc to the right of *Section 1* and end it to the left of *Section 4*. When creating the arc, you can backup by pressing the *Backspace* key. To abort the arc and start over, press the *Esc* key.
3. End the arc by double-clicking the last point.

When you are finished, you will see a centerline arc with arrows pointing upstream, as shown in Figure 15-2.



Figure 15-2 Creation of centerline

### 15.3.2 Defining Cross Sections

---

Cross section arcs can be created across the river. *WSPRO* uses cross sections to compute the geometry of the river and to perform the flow analysis. A feature arc needs to be created at each of the four cross sections shown on the tiff image. To create the cross section arcs:

1. Choose the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create an arc along *Section 1*, making sure to click where the cross section intersects the centerline arc. Double-click the last point to end the arc.
3. Repeat to create the other three cross sections.

When *SMS* converts feature arcs to a *WSPRO* model, one cross section point is created for each vertex. When more vertices are defined along a cross section arc, there will be more detail in the cross sections of the model. The vertices can be redistributed along the arcs by defining a vertex spacing. To redistribute the vertices:

1. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
2. Click on one of the arcs at *Section 1* to select it. The cross section will be highlighted.
3. Select *Feature Objects | Redistribute Vertices*. The *Redistribute Vertices* dialog shows information about the number of segments and segment spacing for the selected arc.
4. Choose the *Number of Subdivisions* option and enter a value of 10. Then click the *OK* button.

The vertices along the selected arc become distributed evenly with ten subdivisions in the arc. When creating a model, the vertices along the cross section may not be uniform. In general, there should be more vertices in the main channel to get a better channel definition, and the vertices in the flood plain should be spaced further apart. The channel definition can be increased by manually adding vertices to an arc. To add vertices to the first cross section in the river:

1. Zoom in to where *Section 1* crosses the river.
2. Choose the *Create Vertex*  tool from the *Toolbox*.
3. Add a few more vertices near the river by clicking along the cross section.

A vertex is added at each point on the cross section where the mouse is clicked. When creating a model, this should be repeated for each cross section until the desired spacing is defined. However, to save time, and to assure continuity, this has

been done for you. The final set of arcs has been saved to a *map* file. To open the map file:

1. Select *File | Open*.
2. Open the file named *redfox\_a.map* from the *tutorial* directory. The map file contains arcs as shown in Figure 15-3.

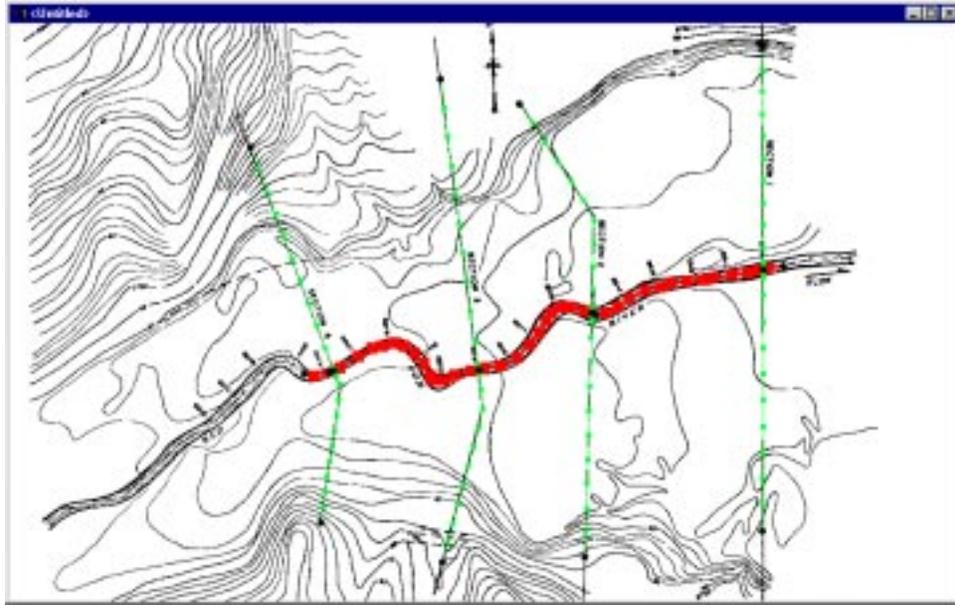


Figure 15-3 The *redfox\_a.map* data.

### 15.3.3 Creating an Area Property Coverage

An *Area Property* coverage uses feature arcs and polygons to divide the area into zones of similar land types. Example land types include river, farmland, brush, and woodland. Different land types affect the river's flow due to the type of material through which the water flows. To create an area property coverage:

1. Select *Feature Objects | Coverages*.
2. Click the *New* button to create a new coverage and enter the name *Land Use*.
3. Change the *Coverage type* to *Area Property*.
4. Click the *OK* button to exit the *Coverages* dialog.

The new coverage will become active and the other coverages will dim. Material zones need to be created in this new coverage. To do this:

1. Choose the *Create Feature Arcs*  tool from the *Toolbox*.

2. Create a box around the centerline and the cross section arcs to define the land area. Be sure to enclose the entire centerline and all of the cross section arcs inside the box.
3. Create arcs to separate the different land types. Outline material sections as shown in Figure 15-4.

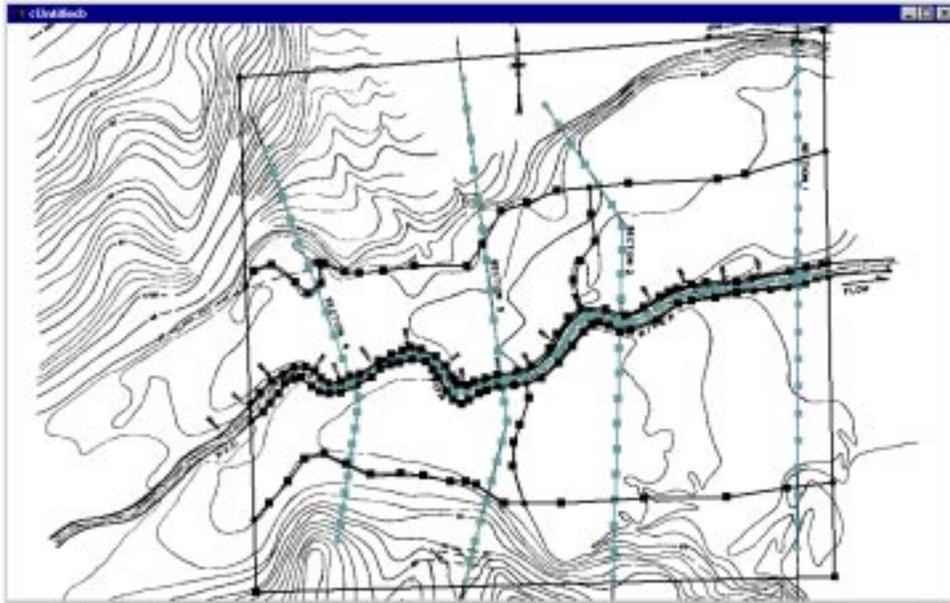


Figure 15-4 The land area boundaries.

With the arcs defined, polygons can be created to define the material zones. To create the polygons from the arcs:

1. Select *Feature Objects / Clean*. Accept all default values and click the *OK* button. This assures that polygons are ready to be built. You should always clean the feature objects before building polygons.
2. Select *Feature Objects / Build Polygons*. At the prompt, click the *OK* button to use all feature arcs.

All material zones should now be defined by polygons. These polygons can be selected using the *Select Polygon*  tool. Before continuing, you should open another map file to make sure your data matches that in this tutorial. To do this:

- Open the file *redfox\_b.map*.

### 15.3.4 Assigning Material Types

With the polygons built, a material type can now be assigned to each polygon. To assign the material:

1. Choose the *Select Polygon*  tool from the *Toolbox*.
2. Double-click inside the polygon that includes the river channel.
3. In the *Land Poly Atts* dialog, select the *Material* option and then click the button just to the right of it. (This button displays the name of the material that is assigned to the polygon.)
4. In the *Materials Data* dialog, click the *New* button to create a new material type. Change the *Material Name* to *Channel*.
5. Click the *Close* button, then the *OK* button to close both dialogs.
6. Repeat this process for each of the material zones, assigning the materials as shown in Figure 15-5. Create new materials as required.

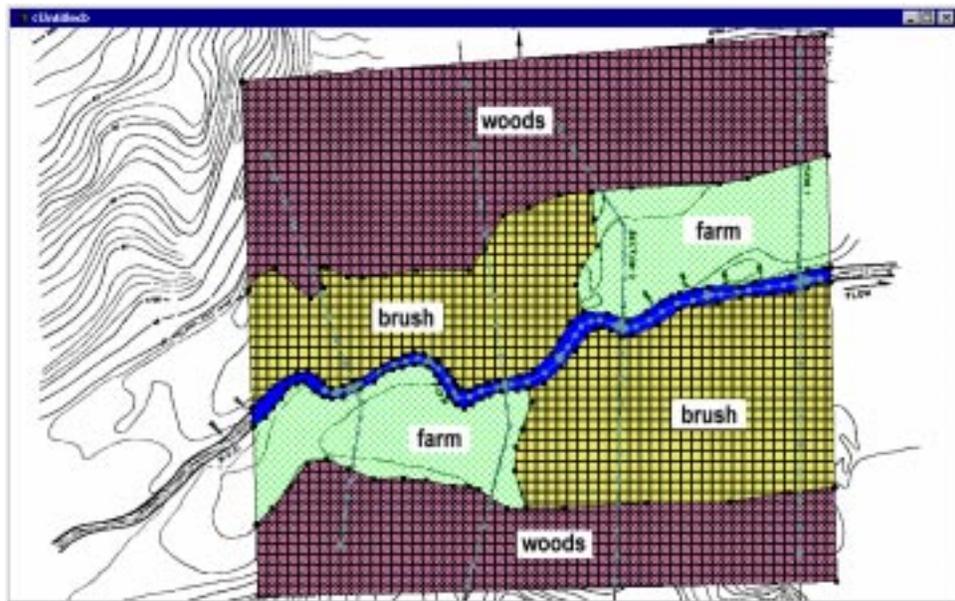


Figure 15-5 Material type boundaries.

### 15.3.5 Importing Topographic Data

The conceptual model defines the locations of the river centerline, the cross sections, and the material zones. However, there is no topographic information for defining elevation data. Topographic data can be taken from scattered data points, an existing finite element mesh, dxf data, or a digital elevation model. For this example, we will take the elevations from an existing finite element mesh which overlaps the model area. To open the finite element mesh:

1. Select *File / Import*.

2. Choose the *2-D Mesh* option and click the *OK* button.
3. Open the file *redfox.2dm*.

The display will refresh, with the triangulated network on top of the feature objects, as shown in Figure 15-6. This mesh was defined by digitizing the contour data from the tiff image. It will provide the topographic information that is required to generate cross sections for *WSPRO*.

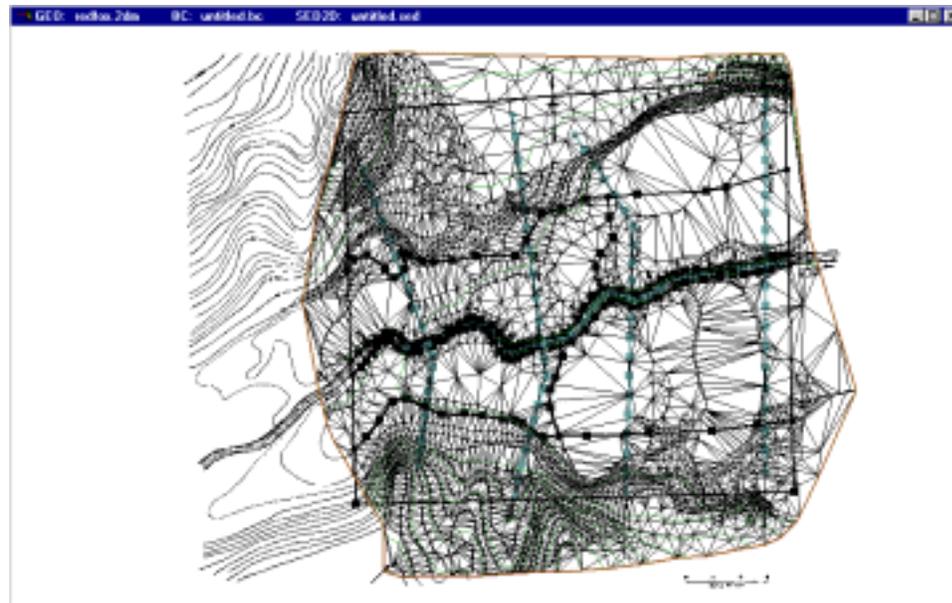


Figure 15-6 The finite element mesh on top of the map data.

## 15.4 Saving the Data

---

The data can be saved altogether with a super file. This keeps all of the data in one location for opening in a later session. To save all the data:

1. Select *File / Save*.
2. In the *Prefix for all files* field, enter the name *redfox1*.
3. Click the *Update* button, then click the *Save* button.

## 15.5 Converting The Conceptual Model

---

Before creating the *WSPRO* model from the conceptual model, you should open the *River* and *Plot* windows. The *River Window* is used to display a one-dimensional

tree representation of the data. The *Plot Window* is used to display cross section and river profile plots. To open these windows:

1. Select *Display / Plot Window*. The screen will split to display the *Plot Window*.
2. Select *Display / River Window*. The screen will split once again to display the *River Window*.

With all three windows open, you are ready to generate the *WSPRO* model. *SMS* will use the topographic data and the conceptual model to create a one-dimensional *WSPRO* model. To create the *WSPRO* model:

1. Selecting *Feature Objects / Coverages*.
2. Highlight the coverage named *oned\_a* and click the *Active* button. Then click the *OK* button. This activates the *WSPRO* coverage that was created earlier.
3. Select *Feature Objects / Map->River*.
4. In the *Domain Options* dialog, make sure the *Use Area Property Coverage* option is on, select the *Land-useb* coverage, and click the *OK* button.

This command converts the 2D data to a 1D *WSPRO* model. A symbol appears in the *River Window* to represent each cross section, and the extrapolated cross sections are drawn in the *Plot Window*, as shown in Figure 15-7. The color bar displayed at the bottom shows the material zones for the active cross section.

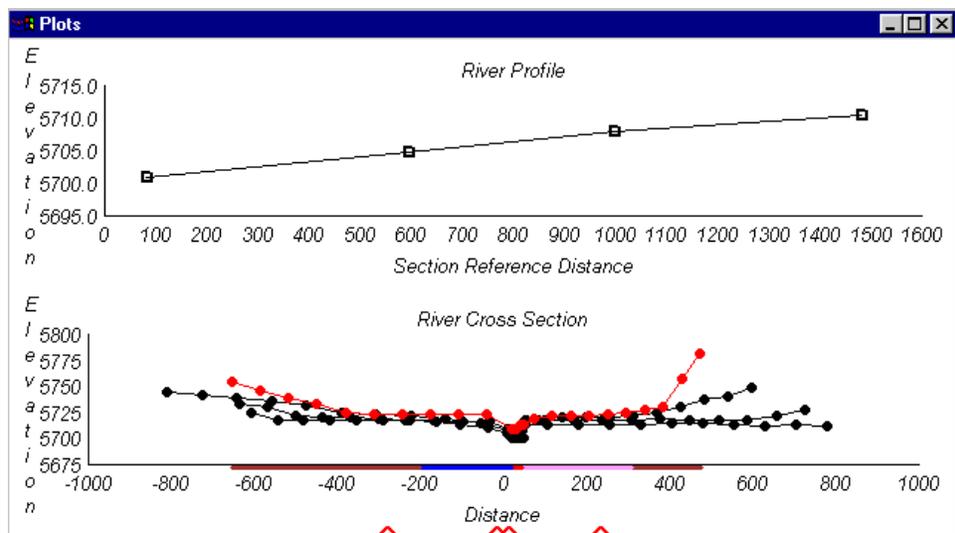


Figure 15-7 The plots in the Plot Window.

## 15.6 Defining The WSPRO Data

---

With the cross sections created, the boundary conditions can be defined for the *WSPRO* model. To access the *WSPRO* data commands:

- Switch to the *River*  module.

### 15.6.1 Assigning Material Values

---

The material zones were automatically created and assigned from the *Area Property* coverage that was built. The locations of these materials and their corresponding *SA* points were applied to the cross sections when the river model was created. Although the material zone locations have been assigned to the *WSPRO* model, the material properties have not. To define the material properties:

1. Select *WSPRO / Roughness Parameters*.
2. Assign the following *Manning's n* values for the materials: *river=0.025*, *woods=0.20*, *brush=0.05*, and *farm=0.06*.

### 15.6.2 Defining The Control Parameters

---

The model control parameters such as flow and solution type must be supplied for *WSPRO* to compute a profile. To assign these parameters:

1. Select *WSPRO / Run Control*. The cross section with the smallest section reference distance (SRD) value will always be included in the *Flow Sets* area. This is the farthest downstream cross section, and a flow rate at this section is required.
2. Click the *Add* button to create a profile.
3. Assign a *Q* (flow) of 5500 cfs to the profile.
4. Assign a *Friction slope* of 0.002 ft/ft, and select the *Subcritical* option.
5. Click the *OK* button to close the *WSPRO Run Control* dialog.

## 15.7 Saving The Simulation

---

Now that the river model has been defined, it is ready for a *WSPRO* analysis. Before the analysis can be run, the simulation must first be saved. To save the simulation:

1. Select *WSPRO / Save Simulation*.

2. Save the simulation as *redfox.dat*.

## 15.8 Running WSPRO

---

*WSPRO* calculates all water surface profiles that have been defined. For this example, only one profile was defined. To run *WSPRO*:

1. Select *WSPRO / Run WSPRO*. A prompt appears to show the location of the *WSPRO* executable.
2. If the *WSPRO* executable is not found, or if *SMS* finds an older version that you don't want to use, click the *File Browser*  button to manually select the proper *WSPRO* executable.
3. Click the *OK* button to run *WSPRO*.

## 15.9 Viewing the Results

---

*WSPRO* creates two output files. The *.plt* file contains the solution results. These results can be viewed inside *SMS* by importing the file through the *Data Browser*. The *.lst* file is a text file that stores tables of solution variables. It can be viewed in any text editor such as *notepad.exe* or *vi*. For more information on viewing the solution results, see lesson 14.

## 15.10 Conclusion

---

This concludes the Advanced *WSPRO* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



---

## ***Observation Coverage***

---

### **16.1 Introduction**

---

An important part of any computer model is the verification of results. Surface water modeling is no exception. Before using a surface water model to predict results, the model must successfully simulate observed behavior. Calibration is the process of altering model parameters until the computed solution matches observed field values within an acceptable level of accuracy. *SMS* contains a suite of tools in the *Observation Coverage* to assist in the model verification and calibration processes.

The observation coverage consists of *Observation Points* and *Observation Arcs* which help analyze the solution for a model. Observation points can be used to verify the numerical analysis with measured field data and calibration. They can also be used to see how data changes through time. Observation arcs can be used to view the results for cross sections or river profiles. This tutorial is based on a *FESWMS* finite element model, but the verification tools in *SMS* can be used with any model.

---

### **16.2 Opening the Data**

---

To open the *FESWMS* simulation and solution data, *SMS* must be in the *FESWMS* portion of the *Mesh Module*. To do this:

1. Switch to the *Mesh*  module.

2. Select *Data / Switch Current Model* and select the *FESWMS* option. If you do not have the *FESWMS* interface registered, then you first must change to *Demo Mode* before being able to select this option. See the *SMS Reference Manual* for more information on this.
3. Click the *OK* button.

Now that *SMS* is using the *FESWMS* interface, the data for this tutorial can be opened. To open the data:

1. Select *FESWMS / Open Simulation*.
2. Open the file *observe1.fil* from the *tutorial* directory. If geometry data is still open from a previous tutorial, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

The mesh data shown in Figure 16-1 will be opened into *SMS*.

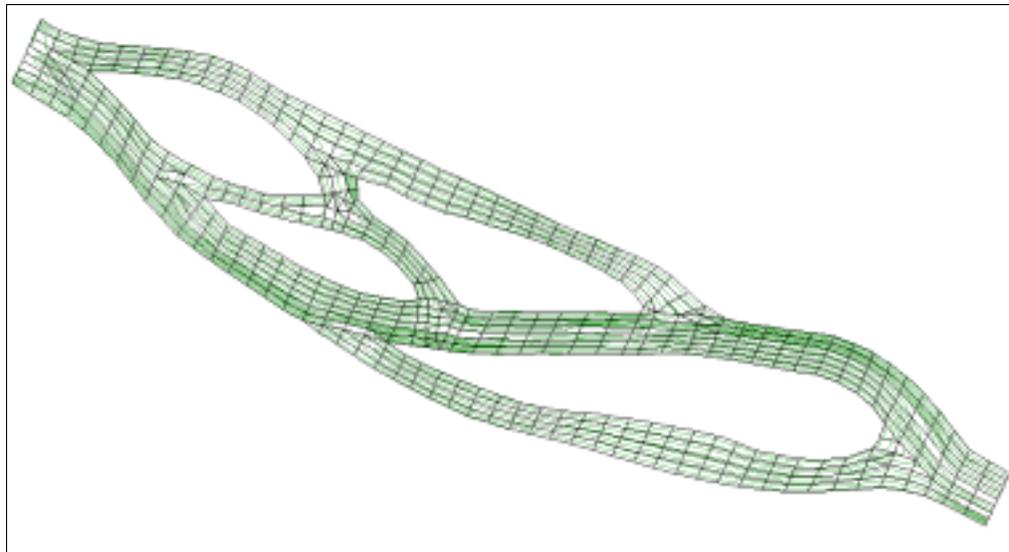


Figure 16-1 The mesh contained in *observe1.fil*.

## 16.3 Viewing Solution Data

---

An initial solution has already been created with this data file. To open the solution:

1. Select *Data / Data Browser*.
2. Click the *Import* button and choose the *FESWMS file* option. Click *OK*.
3. Open the file *observe1.flo*.

4. Select the *Done* button to exit the *Data Browser*.

When the solution file is opened into *SMS*, various scalar and vector data sets are created. By default, the active data sets are the *Velocity mag* scalar data set and the *Velocity* vector data set. Several display options should be changed. To do this:

1. Select *Display / Display Options*.
2. Turn on the *Contours* and *Mesh boundary* options. Turn off the *Nodes* and *Elements* options.
3. Click the *OK* button to exit the *Display Options* dialog.

## 16.4 Creating an Observation Coverage

---

Before creating observation, an *observation coverage* must exist. To create the observation coverage:

1. Switch to the *Map*  module.
2. Select *Feature Objects / Coverages*.
3. Click the *New* button to create a new coverage.
4. Change the *Coverage type* to *Observation*, and change the name of the new coverage to "Observation".

At this point, an observation coverage has been created. The *Observation Coverage Options* dialog is used to define the data associated with observation points. To change the observation coverage options:

1. Click the *Options* button under the *Coverage type*.
2. In the *Measurement Type* section of the *Observation Coverage Options* dialog, click the *New* button to create a new type of measurement.
3. Enter *Velocity Magnitude* for the name.
4. Click the *OK* button in both dialogs.

## 16.5 Creating an Observation Point

---

Observation points are created at locations in the model where the velocity and/or head have been measured in the field. The measured values will be compared with

the values computed by the model to determine the model's accuracy. Each observation point is assigned the following data:

- *Location.* The x, y real world location of the point needs to be specified. The point can be created interactively and then the coordinates can be changed.
- *Function.* The functional value is the value that was measured in the field. This value can be any real data, such as velocity or water surface elevation.
- *Interval.* The interval is the allowable error ( $\pm$ ) between the computed value and the observed value. Model verification is achieved when the error is within the interval ( $\pm$ ) of the observed value.
- *Confidence.* The confidence value represents the percent of confidence that exists in the error estimate.

Table 16-1 Observation point values.

| x [ft] | y [ft] | Velocity [fps] | Interval [fps] | Confidence [%] |
|--------|--------|----------------|----------------|----------------|
| 190    | -369   | 3.5            | .25            | 95             |

One observation point should be created using the values in Table 16-1. In this case, the model will be verified if the computed value is  $\pm 0.25$  fps of the observed velocity, or between 3.25 and 3.75 fps. To create the observation point:

1. Choose the *Create Points*  tool from the *Toolbox*.
2. Click anywhere in the *Graphics Window*. This creates an observation point at the cursor location.
3. In the *Edit Window* at the bottom of *SMS*, change the *X Coordinate* to 190.0 and the *Y Coordinate* to -396.0.

The observation point moves to the specified location (you may have to press the *TAB* key to make the point move). Now that the point is in the correct location, it needs to be assigned the observed value. To assign the attributes to the point:

1. Select *Feature Objects / Attributes*.
2. Enter *Point #1* for the name of the point.
3. Turn on the *Observed* option. This allows observed data to be compared with values computed by the model.
4. Enter 3.5 for the *Observed value*, 0.25 for the *Interval*, and 95 for the *Conf*.
5. Click the *OK* button to close the *Feature Point Attributes* dialog.

### 16.5.1 Using The Calibration Target

A calibration target is drawn next to the observation point. The components of a calibration target are illustrated in Figure 16-2. These components are:

- *Target Middle.* This is the target value that was measured in the field.
- *Target Extents.* The top of the target represents the target value plus the interval while the bottom represents the target value minus the interval.
- *Color Bar.* The color bar shows the error between the observed value and the computed value. If the bar is entirely within the target, the color bar is drawn in green. If the error is less than twice the interval, the bar is drawn in yellow. A larger error will be drawn in red.

For this example, the bar would be green if the computed value is between 3.25 and 3.75, yellow for values between 3.0-3.25 or 3.75-4.0, and red for values smaller than 3.0 or greater than 4.0.

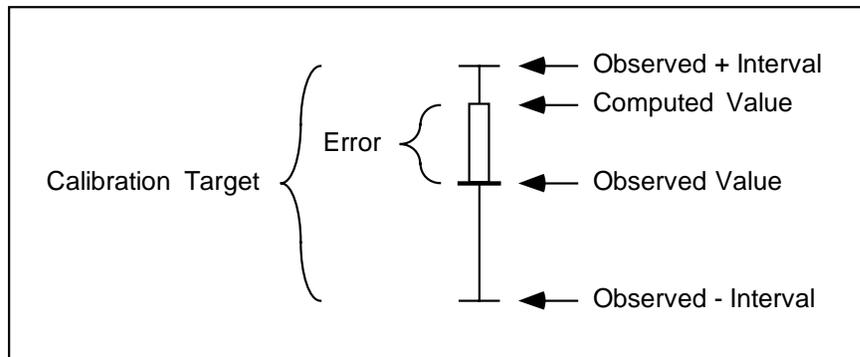


Figure 16-2 Calibration Target.

Now that the observation point has been created and a solution has been opened, the target appears. The color bar in this example is red with an arrow pointing down, indicating that the computed solution has a velocity below 3.0.

## 16.6 Reading a Set of Observation Points

Using the steps defined above, multiple observation points can be created. However, in the interest of time, a set of points will be read from a file. The *observation points* file was created from a spreadsheet. For information on the observation points file format, see the *SMS Reference Manual*. First, the observation coverage should be deleted, because when the observation points file is opened, a new one will be created. To delete the observation coverage:

1. Select *Feature Objects / Coverages*.

2. In the *Coverages* dialog, highlight the coverage named *Observation* and click the *Delete* button. If asked to confirm, click the *Yes* button.
3. Click the *OK* button to close the *Coverages* dialog.

The observation points file can now be opened. To open the file:

1. Select *File | Import*.
2. Choose the *Observation Pts.* option and click the *OK* button.
3. Open the file named *observpts.prn*.

The observation points file is opened and seven new observation points appear. The new points are distributed around the finite element mesh, as shown in Figure 16-3.

FIG

*Figure 16-3 The observation points created from the file observpts.prn.*

## 16.7 Generating Error Plots

---

An error estimation between the model solution and observed data can be computed using observation points. Plots showing the verification error will be generated. To generate these plots:

1. Select *Display | Plot Options*.
2. Click the *New* button and change the *Type* to *Computed vs. Observed*.
3. Click the *New* button again and change the *Type* to *Error Summary*.
4. Click the *OK* button to close the *Plot Options* dialog.

The plots have been created, but they cannot be seen. All plots are drawn in *SMS* in the *Plot Window*. To open the *Plot Window*:

- Select *Display | Plot Window*.

The *Plot Window* and *Graphics Window* are tiled horizontally. At any time, these windows can be resized and moved. The plots that appear are shown in Figure 16-4.

FIG

*Figure 16-4 The error plots in the plot window.*

### 16.7.1 Using The Computed vs. Observed Plot

---

In the *Computed vs. Observed* plot, a symbol is drawn for each of the observation points. A point that plots on or near the diagonal line indicates a low error. Points far from the diagonal have a larger error. To determine which symbol goes with which point:

- Click on one of the points in the plot. The corresponding feature point becomes highlighted in the *Graphics Window*.

The color of individual observation points can be changed. The color of the point in the *Computed vs. Observed* plot corresponds with the color of the observation point.

### 16.7.2 Using The Error Summary Plot

---

In the *Error Summary* plot, the following three types of error norms are reported:

- Mean Error. This is the average error for the points. This value can be misleading since positive and negative errors can cancel.
- Mean Absolute Error. This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.
- Root Mean Square. This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

## 16.8 Calibrating The Model

---

The values in this solution are not within the calibration targets. The material properties will be changed and then the model will be re-run. The errors through the main channel are negative, indicating that the observed velocities are larger than those computed by the model.

### 16.8.1 Editing The Material Properties

---

These computed velocities can be increased by decreasing the eddy viscosity values. To decrease the eddy viscosity:

1. Switch to the *Mesh*  Module.
2. Select *FESWMS | Material Properties*.
3. Change the eddy viscosity value ( $V_o$ ) to 1.5.

4. Click the *Close* button to close the *FESWMS Material Properties* dialog.

## 16.8.2 Computing a New Solution

---

To compute the new solution:

- Save the simulation as *observe2*.
- Run *flo2dh* on the new simulation.

If you are using *SMS* in *Demo* mode, you will not be able to save the simulation. However, this second simulation has been saved in the *output* directory under the *tutorial* directory. You can open the second simulation if you need to.

## 16.8.3 Reading The New Solution

---

The new *FESWMS* solution should now be opened and the errors associated with it should be plotted. To open the second solution:

1. Select *Data / Data Browser*.
2. Click the *Import* button and choose the *FEWMS file* option.
3. Change the *Suffix* to *2* and click the *OK* button.
4. Open the file named *observe2.flo*. If you are running *SMS* in *Demo* mode, this solution file is in the *solution* directory under the *tutorial* directory.
5. Click the *Done* button to close the *Data Browser* dialog.

The plots in the *Plot Window* will automatically update according to the solution that was just opened.

## 16.8.4 Fine-tuning the model

---

Most of the observation points are now within the verification targets. Those that are not are now too high and that the mean error is now positive. This means that the eddy viscosity was lowered too much and it needs to be raised. To finish the verification process:

- Change the eddy viscosity value to 6.0.
- Save and run a third simulation.

- Open the third simulation, giving it a suffix value of 3. If you are running *SMS* in *Demo* mode, this third simulation can be read from the *solution* directory under the *tutorial* directory.

After this third simulation is opened, all the observation point targets should be within the acceptable intervals.

## 16.9 Using The Error Vs. Simulation Plot

When performing trial-and-error verification, it is often important to keep track of the error trend as new solutions are repeatedly computed. *SMS* provides a special verification plot to simplify this task. To create this plot:

1. Select *Display / Plot Options*.
2. Click the *New* button and change the *Type* to *Error vs. Simulation*.
3. Click the *Options* button.
4. In the *Error Vs. Simulation Options* dialog, turn off all data sets except for *velocity mag 1*, *velocity mag 2*, and *velocity mag 3*. This can be done by highlighting each data set setting the *Display* option under the window.
5. Highlight *velocity mag 1* and click the *Move Up* button to move this data set above *velocity mag 2*. Similarly, make *velocity mag 2* above *velocity mag 3*.
6. Click the *OK* button in both dialogs.

The Plot Window updates, showing the error plots vs. simulation, as shown in Figure 16-5. Notice that the errors decrease as each simulation was performed. If the errors increased, then the model is not improving.

FIG

Figure 16-5 The error vs. simulation plot.

## 16.10 Generating Observation Profile Plots

Observation profile plots are used to cut a section in the mesh. These are created along observation arcs in the observation coverage. The first observation arc to be created represents a profile of the main channel. To create this arc:

1. Choose the *Create Feature Arc*  tool from the *Toolbox*.

2. Create an arc down the main channel, as shown in Figure 16-6. Remember to double-click the last point to end the arc.

When the plots are drawn in the *Plot Window*, they will use the name and color associated with the observation arc. To change the name and color of the arc:

1. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
2. Double-click on the profile arc.
3. In the *Observation Arc Attributes* dialog, change the name of the arc to *river profile* (leave its color black).
4. Click the *OK* button to close the *Observation Arc Attributes* dialog.

Three more arcs need to be created, each across a section of the river. These arcs will be used to look at cross section plots. To create these arcs:

1. Select the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create each of the cross section arcs, as shown in Figure 16-6. Note: When creating these, do not click ON the profile arc, as this would split it.

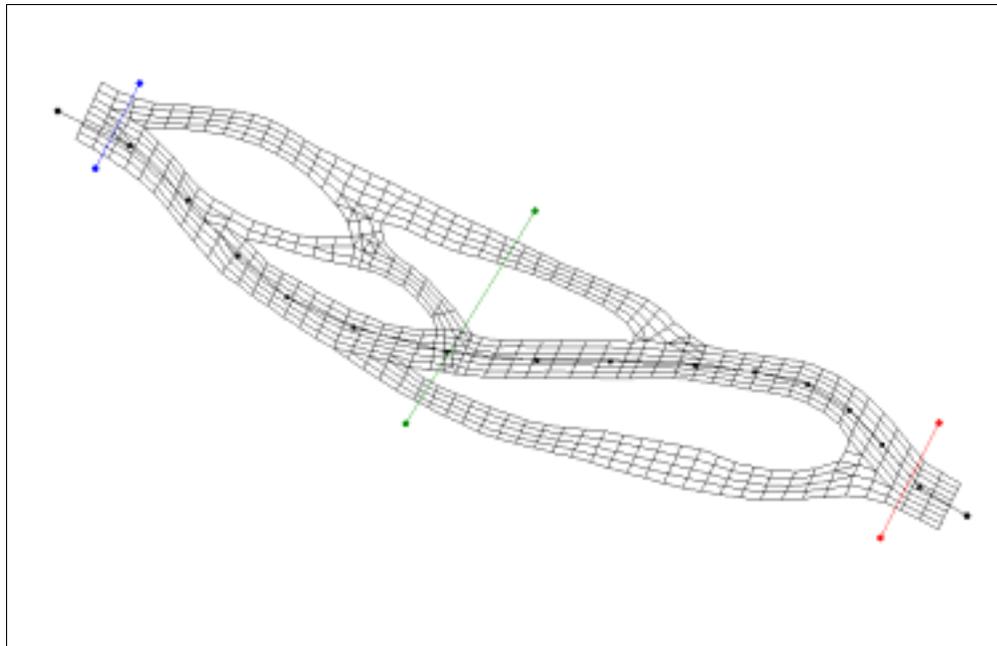


Figure 16-6 Profile and cross-section arcs

With the cross sections created, do the following:

- Select each arc and assign a unique color and an appropriate name.

With the arcs created, the plots can now be generated. To do this:

1. Select *Data / Plot Options*.
2. Click the *New* button and change the *Type* to *Observation Profile*.
3. Turn on the *Use selected data sets* option, and *Display* only the *elevation* and *water surface elevation* data sets.
4. Turn on the *Displayed arcs* option and turn off the three cross-section arcs.
5. Click *OK* in both the ?? and ?? dialogs.
6. Turn off the three previous plots by highlighting each and turning off the *Display plot* option.
7. Under display, be sure that the *Plot*, *Object legend*, and *Curve legend* options are on.
8. Click the *OK* button to exit the *Plot Options* dialog.

The profile plot of the geometry of the stream should be shown in the *Plot Window*. To view the cross sections:

- Create a new cross-section geometry plot. Use the *water surface elevation* and *elevation* data sets, and turn on only the three cross section arcs.

## 16.11 Conclusion

---

This concludes the Observation Coverage tutorial. You may continue to experiment with the program or you can exit *SMS*. To quit *SMS* at this point:

- Select *File / Quit*. If prompted to confirm, click the *Yes* button.



# Index

- 1D Bridges..... 14-3
- 1D River Sections
  - assigning materials ..... 14-6
  - attributes ..... 14-3
  - creating ..... 14-4
  - section break points ..... 14-4
- Adaptive Tesselation ..... 2-10
- ADCIRC
  - analysis ..... 13-5
  - conceptual model ..... 13-1
  - model parameters ..... 13-5
  - saving ..... 13-5
  - solution files ..... 6-1, 13-6
- Animation
  - film loops ..... 6-4
  - flow traces ..... 6-5
  - play controls ..... 6-5
- Arc Group, defining ..... 2-13
- Automatic Mesh Generation ..... 12-6, 13-4
- AVI Files ..... 6-5
- Boundaries, smoothing ..... 3-9
- Boundary Conditions
  - CGWAVE ..... 12-7
  - conceptual model ..... 2-13, 2-14
  - FESWMS ..... 8-3, 8-8
  - HIVEL2D ..... 11-4
  - RMA2 ..... 7-4, 7-8
  - RMA4 ..... 10-4
- Breaklines ..... 3-22, 3-24
- Calibration ..... 16-7, 16-8
- Centerline Arc ..... 15-3
- CGWAVE
  - analysis ..... 12-8
  - boundary conditions ..... 12-7
  - conceptual model ..... 12-4
  - model parameters ..... 12-7
  - saving ..... 12-8
  - setting up SMS ..... 12-1
  - size function ..... 12-3
  - solution files ..... 6-1
  - wavelength function ..... 12-2
- Contours ..... 3-12, 3-20
- Coons Patch ..... 2-11
- Copyright ..... 1-2
- Coverages
  - area property ..... 15-5
  - CGWAVE ..... 12-4
  - manipulating ..... 2-8
  - observation ..... 16-1
  - WSPRO ..... 15-2
- Data Browser ..... 6-2, 16-2, 16-8
- Data Calculator ..... 6-3, 11-4
- Data Sets ..... 6-1
- Demo Mode ..... 1-3, 2-1
- Density Meshing ..... 2-10, 12-5, 13-3
- Disjoint Nodes ..... 3-23
- Display Options ..... 3-13
  - contours ..... 3-12
  - map ..... 12-7
  - materials ..... 2-15
  - mesh ..... 4-2, 8-3, 8-5, 12-7, 16-3
  - scatterpoint ..... 12-7
- Drawing Objects ..... 2-12
- Edit Window ..... 2-3
- Elements
  - adding ..... 3-11
  - defined ..... 3-2
  - deleting ..... 3-3, 3-11
  - editing ..... 3-5
  - refining ..... 3-24
  - selecting ..... 3-3
  - types ..... 3-2
  - types ..... 5-2
- EMS-I ..... 1-2
- Error Plots ..... see *Observation Plots*
- Feature Objects
  - arc ..... 2-5, 2-6, 15-3
  - hide ..... 2-16
  - node ..... 2-5
  - point ..... 2-5
  - polygon ..... 2-5, 12-5
- FESWMS
  - analysis ..... 5-5, 8-9
  - boundary conditions ..... 5-1, 8-3, 8-8
  - conceptual model ..... 5-1
  - hotstart files ..... 8-3, 8-9
  - material properties ..... 5-2, 8-2, 16-7
  - model parameters ..... 5-3, 8-7
  - opening simulations ..... 16-1
  - saving ..... 5-5, 8-7
  - setting up SMS ..... 5-1
  - solution files ..... 5-5, 6-1, 8-10

|                                    |                   |                                 |                 |
|------------------------------------|-------------------|---------------------------------|-----------------|
| weirs .....                        | 8-6, 8-9          | displaying .....                | 11-2            |
| Files                              |                   | editing parameters .....        | 14-6            |
| formats .....                      | 3-1               | Menu Bar .....                  | 2-3             |
| importing .....                    | 3-1               | Mesh                            |                 |
| saving .....                       | 3-16              | editing .....                   | 2-17            |
| Film Loops .....                   | 6-4               | generating .....                | 2-16            |
| Finite Element Mesh                |                   | interpolation .....             | 2-17            |
| boundary .....                     | 3-9               | Meshing Parameters .....        | 2-10            |
| editing .....                      | 3-11              | Model Checker                   |                 |
| generating .....                   | 3-1, 3-2, 3-16    | FESWMS .....                    | 5-4             |
| generating .....                   | 12-6, 13-4        | HIVEL2D .....                   | 11-6            |
| mesh quality .....                 | 3-13, 3-14, 3-15  | RMA2 .....                      | 4-3, 7-6        |
| refining .....                     | 3-21              | Modules .....                   | 1-3             |
| renumbering .....                  | 3-11, 3-15        | Nodes                           |                 |
| renumbering .....                  | 5-2               | adding .....                    | 3-11            |
| saving .....                       | 3-16              | defined .....                   | 3-1             |
| FLO2DH .....                       | see <i>FESWMS</i> | deleting .....                  | 3-11            |
| analysis .....                     | 8-8               | disjoint .....                  | 3-23            |
| setting up SMS .....               | 8-1               | duplicate nodes tolerance ..... | 11-6            |
| solution files .....               | 8-8               | editing elevation .....         | 3-14            |
| Flow Trace .....                   | 6-5               | Nodestrings                     |                 |
| Froude Number .....                | 6-3               | breaklines .....                | 3-22            |
| Gathering Cross Section Data ..... | 3-16, 3-19        | creating .....                  | 3-9, 7-2, 11-2  |
| GFGEN .....                        | 4-3, 7-7          | rectangular patches .....       | 3-16            |
| Graphics Window .....              | 2-2               | renumbering .....               | 3-11, 5-2       |
| Help .....                         | 2-3               | Observation Arcs .....          | 16-1, 16-9      |
| HIVEL2D                            |                   | Observation Coverage            |                 |
| boundary conditions .....          | 11-4              | creating .....                  | 16-3            |
| hotstart files .....               | 11-5              | defining measurements .....     | 16-3            |
| material properties .....          | 11-2              | deleting .....                  | 16-5            |
| model parameters .....             | 11-3              | Observation Plots               |                 |
| saving .....                       | 11-5              | computed vs. observed .....     | 16-7            |
| setting up SMS .....               | 11-1              | cross section plots .....       | 16-10           |
| solution files .....               | 6-1, 11-6         | error plots .....               | 16-6            |
| Hotstart Files                     |                   | error summary .....             | 16-7            |
| FESWMS .....                       | 8-3, 8-9          | error vs. simulation .....      | 16-9            |
| HIVEL2D .....                      | 11-5              | profile plots .....             | 16-9            |
| RMA2 .....                         | 7-4, 7-8          | Observation Points              |                 |
| Image                              |                   | calibration target .....        | 16-5            |
| import .....                       | 2-3               | creating .....                  | 16-3            |
| importing .....                    | 15-1              | file .....                      | 16-5            |
| register .....                     | 2-3               | moving .....                    | 16-4            |
| registering .....                  | 15-2              | Overview .....                  | 1-1             |
| resampling .....                   | 2-4               | Plot Window .....               | 2-2, 15-8, 16-6 |
| Importing                          |                   | Polygons                        |                 |
| mesh file .....                    | 7-1, 11-1         | assigning materials .....       | 2-15            |
| observation points .....           | 16-6              | defining .....                  | 2-9             |
| solution files .....               | 6-2               | Post Processing .....           | 6-1             |
| survey data .....                  | 12-1              | Printing in demo mode .....     | 1-3             |
| Introduction .....                 | 1-1               | Rectangular Patches .....       | 3-17, 3-21      |
| Macros .....                       | 2-3               | defined .....                   | 3-16            |
| Materials                          |                   | options .....                   | 3-17            |
| creating .....                     | 14-5              | Refine Point .....              | 2-10            |

- Refreshing The Display ..... 3-11
- Registration Points..... 2-4
- Renumbering ..... 3-11, 5-2
- River Window ..... 2-2, 15-8
- RMA2
  - analysis ..... 4-4, 7-7, 7-9
  - boundary conditions ..... 4-1, 7-4, 7-8
  - conceptual model ..... 4-1
  - hotstart files ..... 7-4, 7-8
  - material properties..... 4-2, 7-2
  - model parameters ..... 7-3, 7-8
  - REV cards ..... 7-6
  - saving ..... 4-3, 7-6, 7-9
  - solution files ..... 4-4, 6-1, 7-9
- RMA4
  - analysis ..... 10-6
  - creating an input file..... 10-2
  - model parameters ..... 10-3, 10-4
  - point load ..... 10-2, 10-5
  - salinity intrusion ..... 10-5
  - solution files ..... 6-1
- Save
  - ADCIRC simulations ..... 13-5
  - CGWAVE simulations ..... 12-8
  - demo mode ..... 1-3
  - FESWMS simulation..... 5-5
  - FESWMS simulations ..... 8-7
  - Finite Element Mesh ..... 3-16
  - HIVEL2D simulations..... 11-5
  - RMA2 simulations ..... 4-3, 7-6, 7-9
  - simulations ..... 2-19
  - superfiles ..... 2-18
  - superfiles ..... 15-8
  - WSPRO simulations..... 14-7, 15-10
- Scatter Points
  - data browser ..... 12-4
  - from file..... 13-1
  - from mesh nodes ..... 12-3
- SED2D-WES
  - solution files ..... 6-1
- Solution Files..... 3-11
- Starting SMS ..... 2-1
- Switching Numerical Model ..... 5-1, 8-1, 12-1, 16-2
- Thin Triangles ..... 3-3
- TIFF Images ..... 15-1
- Toolbox ..... 2-2
- Tools ..... 1-3
  - create 1D bridges ..... 14-9
  - create feature arcs ..... 16-9
  - create feature points ..... 16-4
  - create nodes ..... 3-22
  - Create Nodestring ..... 3-9
  - create nodestrings ..... 3-22, 7-2, 8-3, 11-2
  - create river sections ..... 14-2
  - create section breaks ..... 14-4
  - create sections ..... 14-4
  - Dynamic ..... 2-3
  - Select Elements..... 3-3
  - select feature arcs..... 16-10
  - select feature polygons..... 12-6
  - select feature vertices ..... 12-5
  - select material regions ..... 14-6
  - select nodes..... 8-6
  - Select Nodes ..... 3-14
  - select nodestrings ..... 3-22, 8-4, 11-4
  - Select Nodestrings ..... 3-10
  - select section breaks ..... 14-5
  - select section points ..... 14-4
  - Split / Merge ..... 3-6
  - Static ..... 2-2
  - Swap Edge ..... 3-7
- Training..... 1-1
- Triangles
  - merging automatically ..... 3-4
  - merging manually ..... 3-6, 3-8
  - thin triangles ..... 3-3
- Vertices, redistributing ..... 2-8
- WSPRO
  - analysis ..... 14-7, 15-11
  - bridges ..... 14-9, 14-13
  - conceptual model ..... 15-2
  - creating river sections ..... 14-2
  - display options ..... 14-7, 14-9
  - flow profiles ..... 14-7, 15-10
  - material properties ..... 14-6, 15-10
  - model parameters ..... 14-7
  - saving ..... 14-7, 15-10
  - setting up SMS..... 14-1
  - solution files..... 14-8, 14-11, 14-12
  - SRD ..... 14-2
- XY Series Editor ..... 7-5