**US Army Corps
of Engineers**
Hydrologic Engineering Center

HEC-FIA
Flood Impact Analysis

User's Manual

Version 2.2
September 2012

CPD-81

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to the Department of Defense, Executive Services and Communications Directorate (0704-0188). Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.
PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ORGANIZATION.

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
Sept 2012	Computer Program Documentation	

4. TITLE AND SUBTITLE	5a. CONTRACT NUMBER
HEC-FIA	
Flood Impact Analysis User's Manual	5b. GRANT NUMBER
Draft	
Version 2.2	5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)	5d. PROJECT NUMBER
CEIWR-HEC	5e. TASK NUMBER
	5F. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	8. PERFORMING ORGANIZATION REPORT NUMBER
US Army Corps of Engineers Institute for Water Resources Hydrologic Engineering Center (HEC) 609 Second Street Davis, CA 95616-4687	CPD-81

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

12. DISTRIBUTION / AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT
The HEC-FIA program enables users to calculate flood damage and benefit accomplishment attributed to flood control projects. The analysis period can be the duration of a single flood event or a longer period of record. The program calculates agricultural damage, urban damage, and benefits for use after an event or annually and summarizes the results in reports. HEC-FIA can also look at dam failure events and evaluate the consequences to support risk assessments for dams. HEC-FIA uses Simplified LIFESim to evaluate the impacts to human life.

15. SUBJECT TERMS
flood damage, project benefit accomplishment

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			
U	U	U	UU	352	19b. TELEPHONE NUMBER

Standard Form 298 (Rev. 8/98)
Prescribed by ANSI Std. Z39-18

HEC-FIA
Flood Impact Analysis

User's Manual

September 2012

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

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

CPD-81

Flood Impact Analysis Package, HEC-FIA, User's Manual

2012. This Hydrologic Engineering Center (HEC) documentation was developed with U.S. Federal Government resources and is therefore in the public domain. It may be used, copied, distributed, or redistributed freely. However, it is requested that HEC be given appropriate acknowledgment in any subsequent use of this work.

Use of the software described by this document is controlled by certain terms and conditions. The user must acknowledge and agree to be bound by the terms and conditions of usage before the software can be installed or used. For reference, a copy of the terms and conditions of usage are included in Appendix H of this document so that they may be examined before obtaining the software. The software described by this document can be downloaded for free from our internet site (www.hec.usace.army.mil).

HEC cannot provide direct user assistance or support for its software to non-Corps users. In the past, for non-Corps users, HEC had provided a list of possible vendors for assistance or support for HEC software. By direction of USACE counsel HEC has discontinued this practice and has removed the list from our web site. Non-Corps individuals should use any internet search engine to locate a vendor that can provide support for the HEC software of interest. However, we will respond to all documented instances of program errors. Documented errors are bugs in the software due to programming mistakes not model problems due to user-entered data.

This document contains references to product names that are trademarks or registered trademarks of their respective owners. Use of specific product names does not imply official or unofficial endorsement. Product names are used solely for the purpose of identifying products available in the public market place.

Microsoft and *Window*s are registered trademarks of Microsoft Corp.

ArcView, ArcGIS Desktop, ArcMap, ArcCatalog, 3D Analyst, and Spatial Analyst are registered trademarks of ESRI (Environmental Systems Research Institute).

HAZUS-MH is a registered trademark of FEMA (US Department of Homeland Security, Federal Emergency Management Agency).

Table of Contents

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

Table of Contents (continued)

Chapters

List of Figures

**Figure
Number**

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

Figure
Number

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

**Figure
Number**

List of Figures (continued)

Figure
Number

List of Tables

**Table
Number**

Chapter 1

Introduction

Welcome to the Hydrologic Engineering Centers' Next Generation Flood Impact Analysis (HEC-FIA) computer program version 2.2. It is the successor to the Project Benefit Accomplishment package, HEC-PBA. HEC-FIA includes damage-analysis capabilities, the ability to analyze geo-referenced gridded data, data connectivity with HEC-DSS, tabular reports, and interfaces with geographic information systems (GIS). It also includes a graphical user interface (GUI) that allows the user to enter and edit data necessary for impact analysis.

1.1   Purpose

The HEC-FIA program provides the capability to estimate the impacts associated with flood events and the benefits attributed to flood control projects. FIA is designed to facilitate:

- Assessment of disaster impacts after a flood particularly with the PL84-99 program in mind.
- Analysis of flood impacts in real time to assist with decision-making and response activities.
- Development of annual reports of Corps' project benefit accomplishments that are required by HQUSACE and Congress.
- Development of consequence estimates to support risk assessments for dam safety.

FIA evaluates impacts either with: (1) geo-referenced data grids with inundation, terrain, agricultural, and structural data or (2) using single, continuous, or forecasted HEC-DSS hydrographs. For the specified analysis period, the program estimates:

- Structure damage
- Agricultural flood damage
- Area inundated
- Number of structures inundated
- Life loss

● Project benefits

FIA includes a number of reports that detail these impacts. The impacts can be sorted by impact area, state, Corps District, county, community, study, subbasin, township, congressional district, or flood control district. Another available report details the damage prevented by the flood control projects identified for the study.

To calculate impacts, FIA requires the user to identify the study, assemble and enter data, and execute the program. These analysis steps are covered in detail in this manual. For a brief overview of FIA and how to build a study quickly, refer to the HEC-FIA Quick Start Guide.

1.2   Organization of Manual

The organization of this manual is shown in Table 1.1. The capabilities of FIA are described in Chapters 3 through 12.

Table 1.1 Organization of HEC-FIA User's Manual.

If you want to find out…	Refer to…
The layout of the FIA GUI	Chapter 2
How to work with layers	Chapter 3
How to enter and edit global data	Chapter 4
How to define watershed elements (stream alignment, impact areas, computation points, projects, configurations, etc.)	Chapter 5
How to upload a digital terrain model	Chapter 6
How to upload inundation data grids (depth grids, arrival time grids, and duration grids)	Chapter 6
How to add and define agricultural data	Chapter 7
How to add and define structure inventories	Chapter 8
How to use modified inventories for Non Structural measures	Chapter 8
How to create alternatives	Chapter 9
How to define a time window	Chapter 10
How to run simulations	Chapter 11
How to view simulation results	Chapter 12
How to prepare data for use in HEC-FIA	Appendix B
How FIA computes project benefits	Appendix C
How to setup coordinate systems	Appendix D
How to edit map layer display properties	Appendix E
How to use the Color Chooser	Appendix F

How to use HEC-DSSVue | Appendix G

1.3   References

This User's Manual, the HEC-FIA Quick Start Guide, Technical Reference Manual, and the HEC-FIA program are available on the Hydrologic Engineering Center's web site (www.hec.usace.army.mil).

Chapter 2

Overview of HEC-FIA

This chapter starts with an introduction to the layout of the HEC-FIA GUI, including display elements, menu bar, and tools. The chapter then gives an overview of the process of creating, opening, and saving a study in HEC-FIA.

2.1   Layout and Organization

Users interact with the GUI through what is called the **Main Window**. To ensure an efficient and logical interaction with GUI, the main window is divided into **windows** and **panes** with distinct functions (Figure 2.1): the **Study Pane**, **Content Pane**, **Desktop Pane**, **Map Window**, and **Message Pane**.
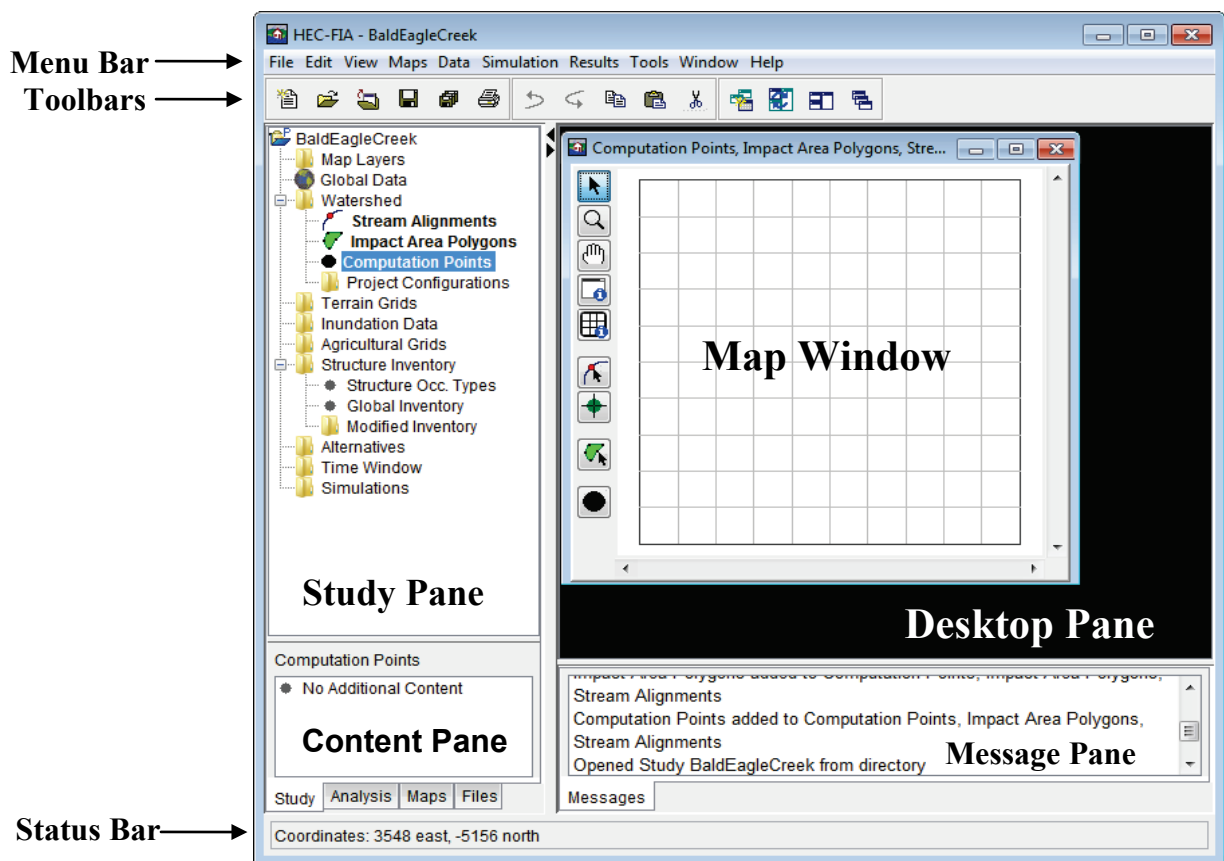
Figure 2.1 FIA Main Window.

2.1.1   Main Window

The **Title Bar** (Figure 2.1) displays the FIA title. After a study is opened, the name of the study will appear here. The **Menu Bar** contains the menus that contain a variety of tools. Many of the functions contained in the menu bar are also available through **shortcut menus** in the **Study Pane** (Section 2.1.4), and will be covered in the respective chapters for each function.

File   This menu is used for study management and contains the following commands: **New Study**, **Open Study, Save Study, Save Study As, Close Study**, **Study Properties**, **Recent Studies**, and **Exit**.

Edit   This menu contains the following edit options, also available using the clipboard **toolbar** of the **main window** (Figure 2.1) or by using their respective keyboard shortcuts: **Undo** (Ctrl+Z), **Redo** (Ctrl+Y), **Cut** (Ctrl+X), **Copy** (Ctrl+C), and **Paste** (Ctrl+V).

View   This menu is used to edit display options of the various FIA windows. The toolbars in the main window (Figure 2.1) may be displayed or hidden. Windows and panes may be displayed or hidden. Users may also save display layouts to use with future studies or restore their current study to a particular layout. Users may also change display units. Available commands include:

Toolbars	displays or hides the standard, clipboard, and maps toolbars.
Study Pane Window	displays or hides the Study Pane
Messages Window	displays or hides the Map Window
Status Window	displays or hides the Status Bar
Toggle Views	toggles between displaying only the Map window
Save Current Layout	stores the current layout for future application
Restore Layout	allows users to apply saved layouts
Layout Manager	shows saved layout
Study Display Units	changes units between English watershed units or SI units.

Maps   This menu has a variety of tools for editing map layer display including: **Default Map Properties**, **New Map**, **Add Map Layers**, **Remove Map Layers**, **Map Window Settings**, **Zoom To Map Extents**, **Save Map Image**, **Import** (stream alignment or impact areas), and **Export** (stream alignment, impact areas, or structure inventories). These functions will be explained in more detail in Chapter 3.

Data   This menu provides access to HEC-FIA data management tools, like the **Global Data Editor** (Chapter 4), the **Impact Area Properties Editor** (Chapter 5, Section 5.2.3, page 5-27), and the **Structure Occupancy Type Editor** (Chapter 8, Section 8.11). It also allows users to upload and import a variety of data. Each of these options will be covered in the portions of this manual devoted to adding data to FIA.

Simulation   This menu contains the tools for creating, editing, and deleting simulations.

Results   Allows users to access a variety of reports after computing simulations.

Tools   This menu provides access to DSS data (HEC-DSSVue), allows creating and editing of scripts, viewing of output logs, viewing of program activity, and allows you to view general information. Available commands are: **HECDssVue, Scripts, Options, Console Output, Customize** and **Memory Monitor**.

Window   This menu provides the display options for the **Map Window** in HEC-FIA. Available commands include:

Tile	tiles all current windows
Cascade	cascades all current windows.
Next Window	scrolls to the next window open in HEC-FIA.
Previous Window	scrolls to the previous window open in HEC-FIA.
Window Selector	opens a dialog box that allows users to select which current windows to display, open, or close.
Window	shows a list of all current windows open in HEC-FIA, pick one to bring it to the front.

These options are also available in the **maps toolbar** in the **toolbars** section of the **main window** (Figure 2.1).

Help   Displays current version information about FIA.

The **Status Bar** (Figure 2.1) displays the coordinates of the pointer on the display area. When the pointer is outside of the display area, the **Status Bar** shows the coordinates of the last location of the cursor in the **Map Window**. The **Message Pane** displays a list of messages that you can scroll through to view system output information.

2.1.2   Toolbars

FIA provides toolbars for quick access to functions that create, edit, and display study elements. There are three toolbars in FIA: the **standard** toolbar, the **clipboard** toolbar, and the **maps** toolbar. Display or hide each toolbar in the **View** menu by pointing to **Toolbars** and selecting the name of the toolbar you wish to display or hide.

The standard toolbar (Figure 2.2) provides the tools to create, open, save, and print study elements. The tools in Figure 2.2 are defined from left-to-right in Table 2.1:
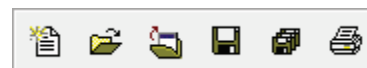
Figure 2.2 Standard toolbar.

Table 2.1 Standard toolbar commands in FIA.

Tool	Description
Create new object	This tool allows users to create a new study, alternative, simulation, configuration, time window, or modified inventory and add inundation data, duration groups, or arrival time groups. Clicking the tool will evoke the **Create New…** dialog box. Creating each of these elements will be covered in their respective chapters.
Open a file	Use to open a file.
Close the study	Closes the current study but leaves FIA open.
Save the study	Saves the current study.
Save everything	Saves the current study and all related data.
Print	Prints the current view of the Map Window.

The **clipboard** toolbar (Figure 2.3) contains common editing tools: (from left to right) **Undo**, **Redo**, **Copy**, **Paste**, and **Cut**.
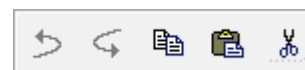
Figure 2.3 Clipboard toolbar.

The **maps** toolbar (Figure 2.4) contains the tool for toggling the display of map windows. The tools are described from left to right in Table 2.2.

Figure 2.4 Maps toolbar.

Table 2.2 Maps toolbar commands in FIA.

Tool	Description
New map	Opens a new map window for the currently open study.
Synchronize	Synchronizes all open map windows to the selected map window.
Tile	Fits all of the open map windows to the area in the desktop pane. Use this tool to all open windows at once.
Cascade	Cascades currently open map windows.

2.1.3   Map Window

The **Map Window** is the area that FIA displays the elements of your study (Figure 2.5). Users can edit and manipulate data in a variety of ways through the use of shortcut menus and map window tools.
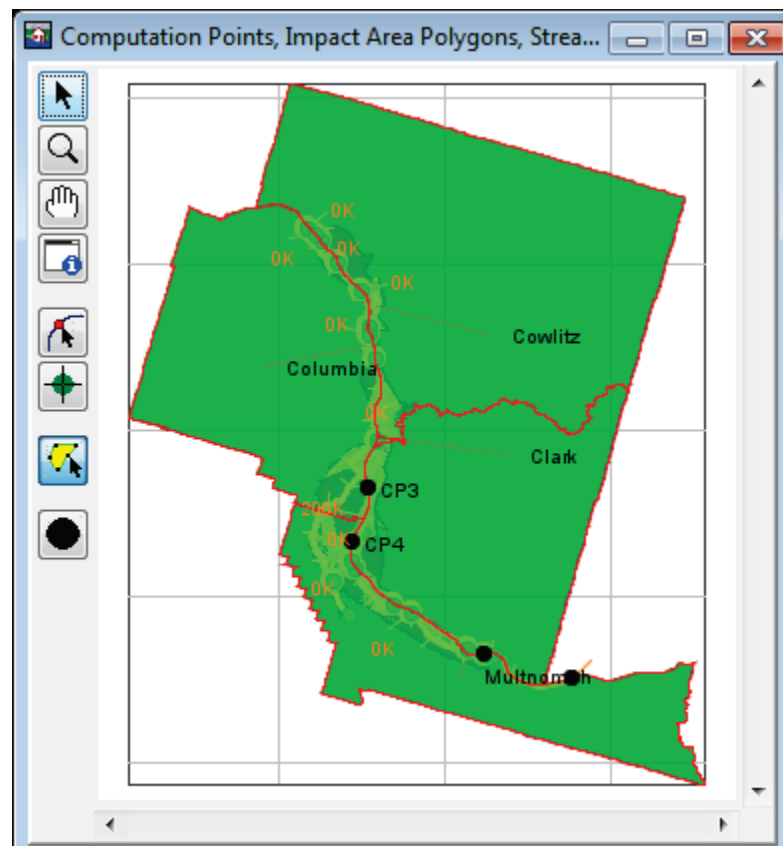
Figure 2.5 FIA Map Window.

Map Window Tools

The following is an overview of available map window tools. If a tool is associated with a particular kind of map layer (e.g. the arrival time tool for arrival time grids), it will only be available when that layer is added to the study. Tools used for creating watershed elements (stream alignments, diversions, channel modifications, reservoirs, etc.) are available whenever a **project configuration** is displayed in your study.

Select Tool – use this tool to edit, rename, and delete elements (does not include stream alignment elements). Commands available for editing existing study elements are available by selecting the **Pointer/Selector Tool**, and right-clicking on a study element.

Zoom Tool - use this tool to zoom in and out of the display area. To zoom in, hold the mouse button down and outline the area you want to zoom in on. To zoom out, click the right mouse button (right-click).

Pan Tool – use this tool to move the display area while you are zoomed in.

Shapefile Query Tool – use this tool to query shapefile information at specific points on a shapefile. This tool is functional when a shapefile is uploaded to your study (Chapter 3, Section 3.1.1, page 3-1).

Stream Alignment Tool - use this tool to create, delete, and edit stream elements that make up the stream alignment (Chapter 5, Sections 5.1.2, page 5-2).

Stream Node Tool - use this tool to create, delete, and edit stream nodes on the stream alignment (Chapter 5, Section 5.1.5, page 5-9).

Impact Area Polygon Tool - use this tool to create, delete, rename, and edit impact area properties. See Chapter 5, Section 5.2.2 (page 5-23) for more detail on creating, deleting, renaming, and editing impact areas.

Grid Query Tool – use this tool to query gridded data. For an inundation grid, a click will display depth values. For an arrival time grid, a click will display the computed arrival time of flood waters during an event for the point at which you clicked. For a duration grid, it will display values of the duration of water during a flood event for the cell where you click. For a digital terrain model, the tool queries elevation values. This tool is only functional when terrain or inundation grids have been added to a study.

Reservoir Tool - use this tool to create, delete, rename, and edit reservoirs. See Chapter 5, Section 5.3.2 (page 5-33) for more detail on creating, deleting, renaming, and editing reservoirs.

Levee Tool - use this tool to create, delete, rename, and edit levees. See Chapter 5, Section 5.4.1 (page 5-41) for more details on creating, deleting, renaming, and editing levees.

Diversion Tool - use this tool to create, delete, rename, and edit diversions. A diversion has to be started on the stream alignment. To create a diversion, select the **Diversion Tool**, hold down the **CTRL** key, and click on the stream alignment. Each click creates a vertex point along the diversion. When reaching the end point of the diversion, release the **CTRL** key, and click. You will be asked to name the diversion. After naming the diversion, the diversion will appear on the display area. Commands available for editing an existing diversion are available by selecting the **Diversion Tool**, and right-clicking on a diversion.

Channel Modification Tool - use this tool to create, delete, rename, and edit channel modifications. A channel modification has to be created on the stream alignment. To create a channel modification, select the **Channel Modification Tool**, hold down the **CTRL** key, click on the upstream end of the channel modification, release the **CTRL** key, and drag along the stream alignment to the downstream end of the channel modification and click. You will be asked to name the channel modification. After naming the channel modification, the channel modification will appear on the display area. Commands available for editing an existing channel modification are available by selecting the **Channel Modification Tool**, and right-clicking on a channel modification.

Off-Channel Storage Area Tool - use this tool to create, delete, rename, and edit off-channel storage areas. To create an off-channel storage area, select the **Off-Channel Storage Area Tool** and hold down the **CTRL** key while clicking the mouse button. Each click creates a vertex point, which is the boundary of the polygon. When reaching the end point, release the **CTRL** key and click. You will be asked to name the off-channel storage area. After naming the off-channel storage area, the off-channel storage area will appear on the display area. Commands available for editing an existing off-channel storage area are available by selecting the **Off-Channel Storage Area Tool**, and right-clicking on an off-channel storage area.

Other Project Tool - use this tool to create, delete, rename, and edit other projects. This tool enables the placement of elements that represent "other" projects, such as pump stations and gages. To create an "other" project, select the **Other Project Tool** and hold down the **CTRL** key and click. You will be asked to name the "other" project. After naming the "other" project, the "other" project will appear on the display area. Commands available for editing existing "other" projects are available by selecting the **Other Project Tool**, and right-clicking on an "other" project.

Impact Area Tool – use this tool to access the **Impact Area Editor**. Engage the tool and right-click on an impact area in your study. From the shortcut menu, select **Edit Impact Area**. The **Impact Area Editor** will open, displaying the detailed information entered for that impact area. For more on the **Impact Area Editor** (which is now part of the process of defining an **alternative**), Chapter 9, Section 9.7 (page 9-8).

Structure Tool – use this tool to create, delete, rename, and edit individual structures. For more on structure inventory editing using the **Structure Tool** (and otherwise), Chapter 8, Sections 8.1.4 (page 8-5), and 8.2 (page 8-5).

Computation Point Tool - use this tool to create, delete, rename, and edit computation points. Computation points are locations where time-series information is to be exchanged between models. For FIA, this would be the location of the hydrograph. See Chapter 5, Section 5.5 (page 5-47) for more detail on creating, deleting, renaming, and editing computation points.

Using Shortcut Menus

Using the previously described **Map Window** tools, users can edit the study elements displayed in the window using shortcut menus. For example, when users engage the **Structure Tool**, they can locate an individual structure they wish to edit, right-click, and perform a variety of actions from a shortcut menu, as shown in Figure 2.6.
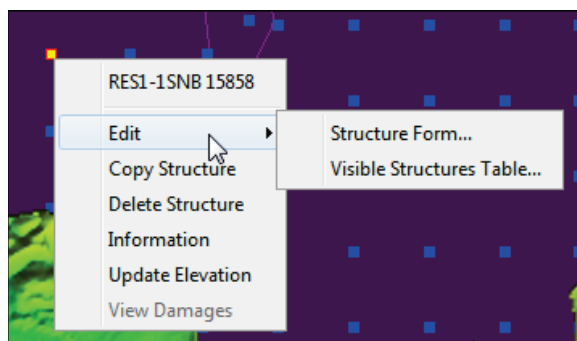
Figure 2.6 Structures shortcut menu.

This makes creating and editing studies in FIA efficient and intuitive. The editing capabilities for different study elements and data will be described in their individual chapters of this manual.

2.1.4   Study Pane

In the **Map Window**, users can interact with their study as it is displayed in a geo-referenced map format. In the **Study Pane**, users interact with a schematic that displays the logical flow of their study in terms of files and folders. This schematic is called the **Study Tree** (Figure 2.7).

The way users interact with the Study Tree is by right-clicking on elements and performing commands from shortcut menus.

In the figure at right, map layer *CRT_Wkshp_Centerline.shp* was uploaded by right-clicking on the **Map Layers** folder and selecting **Add Map Layers** from the shortcut menu. The terrain model *fia_dem.smrmp* was uploaded by right-clicking the **Terrain Grids** folder and selecting **Add Terrain Model**.

Double-clicking on a given map layer or element will display it (if hidden) or hide it (if already displayed) in the map window if it is a displayable element. Bolded map layers in Figure 2.7 are displayed.
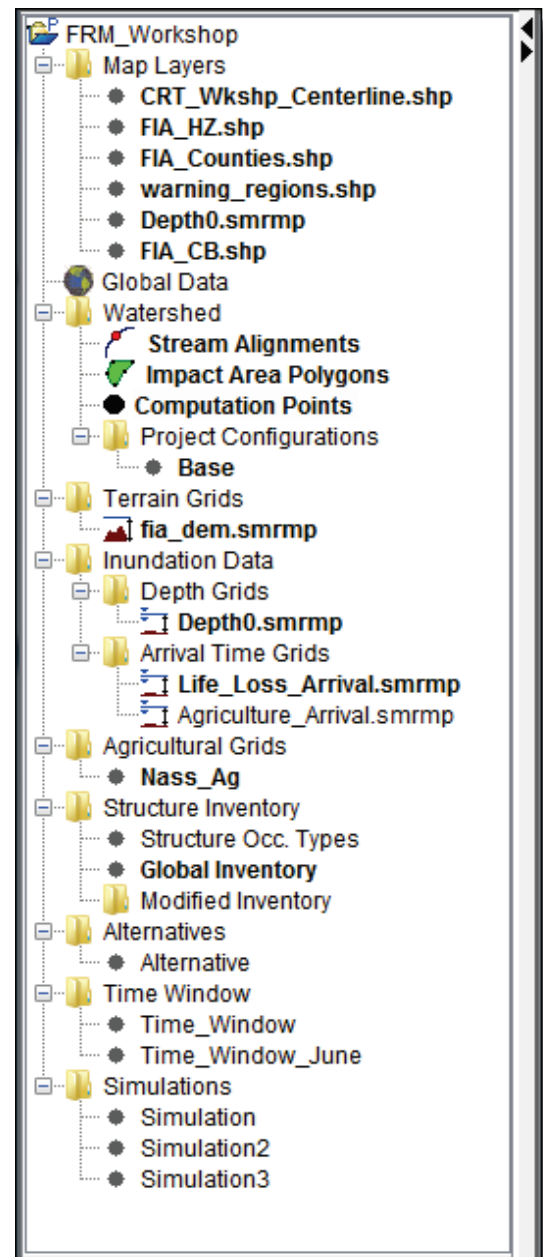
Figure 2.7 Study Pane and example Study Tree.

Many layers can be further defined using editors. These editors are accessible through the shortcut menu for each element. For example, right clicking on the agricultural grid *Nass_Ag* and selecting **Edit** would evoke a dialog for crop loss function definition.

Editing study elements within the **Study Pane** provides a familiar and intuitive approach to study management. The individual functions for editing each study element will be covered in detail in their respective chapters.

2.1.5   Content Pane

The **Content Pane** displays information about study elements that cannot display in the **Map Window** or the **Study Tree**. For example, FIA records the last date each simulation in a study has been computed. When users highlight a computed simulation in the **Study Pane,** this additional information will display in the **Content Pane**. Figure 2.8 displays the last run date and the time window of the simulation *Upstream_Control.*
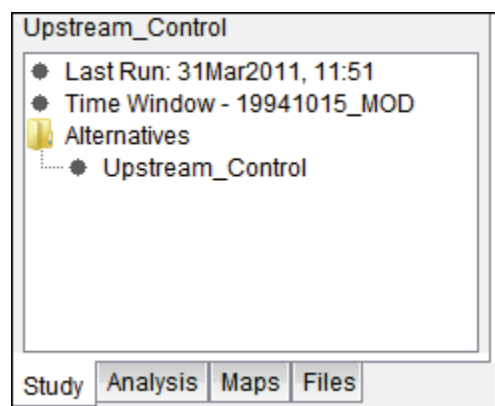
Figure 2.8 Content Pane.

For computed simulations, users may also access consequence reports within the **Content Pane** by right-clicking on computed alternatives (e.g. *Upstream_Control*) and selecting the report from a shortcut menu.

Other study elements may display different data, or have no additional data to display. When users click on **Stream Alignments** in the **Study Tree**, the **Content Pane** will display a list all streams in the study. Clicking on each stream name will highlight that stream in the **Map Window**.

If there is no information to display in the **Content Pane**, it will read *No Additional Content*.

2.1.6   Data Display Tabs

Directly beneath the **Content Pane** are four tabs labeled **Study**, **Analysis**, **Maps**, and **Files** (Figure 2.8). Use these tabs to change the data displayed in both the **Study Pane** and the **Content Pane**.

Study – when this tab is selected, the **Study Pane** will display the **Study Tree** and the **Content Pane** will display additional information about a selected map element. Studies will open on the study tab by default.

Analysis – when this tab is selected, users can group simulations together for analysis. The **Study Pane** will display the simulations within an analysis group and their respective alternatives; the **Content Pane** will display additional information about each simulation (just as in Figure 2.8).

Maps – select this tab to a list of all map layers and elements in your study in the **Content Pane**. Users may toggle map display options, including display order, display properties, and display on/off. The **Study Pane** will display a list of currently opened maps (users may open more than one map of the same study for displaying separate simulations, separate inundation grids, etc.).

Files – select this tab to view a schematic of the directory of your study. In the place of the **Study** and **Content Panes** will appear a list of the folders and files stored on your hard drive for the current study.

2.2   Starting HEC-FIA

To start HEC-FIA, double-click the HEC-FIA program icon on your desktop, or from the taskbar click **Start**, point to **Programs**, point to **HEC**, and then click HEC-FIA. The splash dialog box for FIA will open (Figure 2.9).
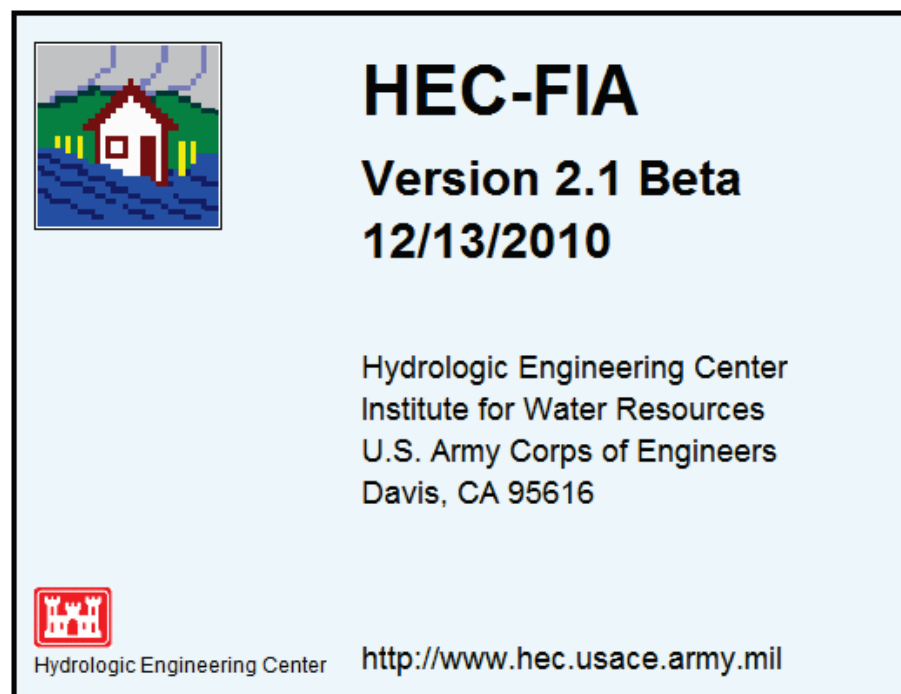
Figure 2.9 HEC-FIA splash dialog box.

2.3   Create a New Study

In FIA, the set of files associated with a particular area is called a **study**. FIA creates a directory (folder) with the study name specified, and stores all the files associated with the study in that directory.

To create a new study:

1.   Open HEC-FIA.
2.   On the **File** menu (Figure 2.1, page 2-1), click **New Study**.
3.   The **Create New Study** dialog box (Figure 2.10) will appear.

Figure 2.10 Create New Study dialog box.

4.   Enter a name (required) and a short description.
5.   The **Directory** field shows you the location in which FIA will create your study directory to store and retrieve data. It will automatically match the name you enter in the **Study Name** field. To choose a different directory location, click the browse ⬚ button and use the **Open Browser** to select a different location in which to create your study directory. In the example in Figure 2.11, FIA will now create the *BaldEagleCreek* study directory in *C:\FIAProjects\BaldEagleCreek*.

Figure 2.11 Open Browser.

6. Select system units for the study from the **Units** list. English and SI units are available. English units will be selected by default.

7. At this time, users may choose to select a **coordinate system** (the projection system HEC-FIA will use for geo-referencing in the **Map Window**), but this is not required.

8. At the bottom of the dialog, click **OK**. The **Create New Study** dialog box will close and a schematic of your study called the **Study Tree** will appear in the **Study Pane** with the new study name at the top (Figure 2.12).
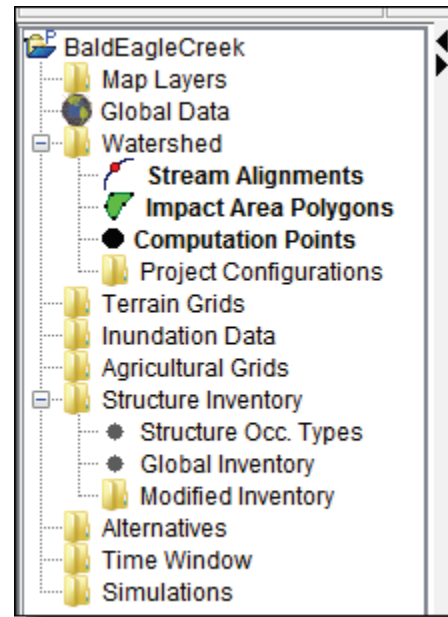
Figure 2.12 Study Tree.

2.3.1   Opening an Existing Study

There are two ways to open existing studies in FIA. One way is to open a study by selecting it from the list of recently-used studies. This list is located at the bottom of the **File** menu under **Recent Studies** and contains the names of as many as six studies most recently opened in FIA (Figure 2.13). Click on the name of the study you wish to open, and FIA will automatically load the selected study. The second way to open an existing study is listed in the steps below.
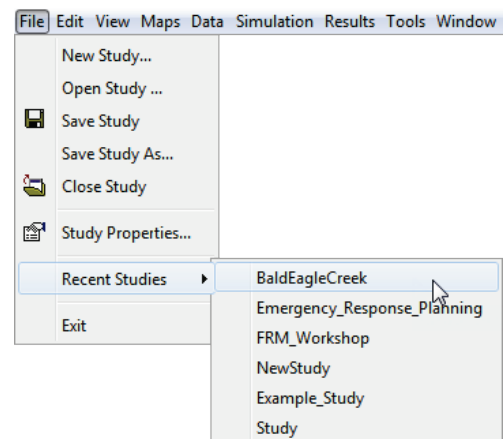
Figure 2.13 File menu—Recent Studies.

To open an existing study:

1. On the **File** menu, click **Open Study.**
2. The **Open Study** dialog box will open (Figure 2.14).
3. Browse to the folder in which you save your FIA project files. In the example in Figure 2.11 (page 2-12), this folder was *C:\FIAProjects*. Within the *FIAProjects* folder, FIA will create a folder for each study you create. In Figure 2.14, the folder *BaldEagleCreek* has been opened. Within this folder are the **study**

directory folders (i.e. folders for studying various study data) and the **project file**. Click on the project file of the study you wish to open (e.g. *BaldEagleCreek.prj*), then click **Open**.
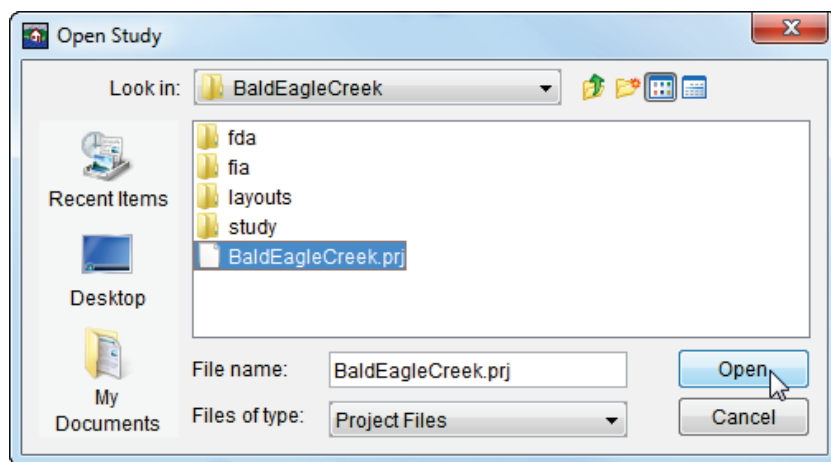
Figure 2.14 Open Study browser.

4.   Your study will now open. The **Study Tree** will display in the **Study Pane** and the **Map Window** will display the contents of your study.

You can only have one study open at a time in FIA. Therefore, if a study is open and you ask FIA to open a different one, FIA will automatically save the current study, close it, and open the new study.

2.3.2   Save a Study

FIA automatically saves study data every time a change is made. To manually save the changes you make to a study, on the **File** menu (Figure 2.13), click **Save Study**. This will save all changes made to the active study. When exiting the program, FIA will automatically save changes.

Chapter 3

Map Layers

3.1   Map Layers Concepts

To help you visualize a system configuration and associated flood impacts, FIA has the capability to display maps of system features. County, Corps district, political, and impact area boundaries can be viewed in the display area. FIA 2.2 also has the ability to retrieve data from maps.

3.1.1   Map Layer Formats

In FIA, digital maps are referred to as map layers. When a map layer is added to the study, the program displays it in the map window. FIA can use certain types of map layers as data sources. The map layer formats that FIA can display are listed in Table 3.1.

Typical data map layers required for a study include:

- Stream centerline shapefile (i.e., *Truman_StreamCenterline.shp*)
- Census block, parcel, or structure point shapefile (i.e., *Truman_ CensusBlock.shp*)
- Impact area shapefile (i.e., *Truman_Counties.shp*)

Typical data map layers used but not required for a study include:

- Hazard area shapefile (Life-Loss) (i.e., *Truman_HZ.shp*)
- Warning regions shapefile (Life-Loss) (i.e., *Truman_WR.shp*)
- Cross Sections shapefile (Life-Loss) (i.e., *Truman_XS.shp*)
- Storage Area shapefile (Life-Loss) (i.e., *Truman_SA.shp*)
- Computation Point shapefile (i.e., *Truman_CCP.shp*)

Table 3.1 Map layer formats compatible with HEC-FIA.

Description	Common Filename Extension
USGS digital line graph	.dlg
AutoCAD® DXF	.dxf
Arc Shapefile	.shp
Raster image	.img, .jpg, .gif, .tiff/.tif
GeoTIFF	.tiff/.tif
ASCII NetTin	.net
USGS DEM	.dem
ArcInfo® DEM	.asc
LizardTech® MrSID®	.sid
HEC-FIA SMRMP	.smrmp

Arc Shapefiles (.shp)

This layer type is the native data structure for ESRI® GIS software.
Shapefiles store non-topological geometry and attribute information for
the spatial features of a data set. Usually there are three files associated
with a shapefile: *.dbf, *.shp, *.prj, and *.shx. The *.shp* file contains the
shapes (point, polylines, or polygons) that are displayed in the

layer. Each shape has a record in the .dbf file containing descriptive data
called "attributes." The .shx file contains an index that links each shape to
its record in the .dbf file. The .prj file contains the projection information
used to define the spatial location of features.

AutoCAD® DXF (.dxf)

A Drawing Interchange File (dxf) is created by a Computer-Aided Design
(CAD) package, AutoCAD®. This format is a tagged data representation
of an AutoCad® drawing file. **Tagged data** is where each data element is
preceded by a group code. A **group code** is an integer indicating the type
of data element. The file created is a vector-based graphic. The DXF
format is commonly used for data exported from other CAD and GIS
programs.

Raster Image

The raster image formats supported by FIA are JPEG format (*.jpg),
Portable Network Graphics format (*.png), Bitmap format (*.bmp),
Tagged Image File Format (*.tiff/*.tif) and Graphics Interchange Format

(*.gif). For FIA to access these file types, an *.img* (ASCII text) file must be created by the user outside of FIA. The file needs to contain the information shown in **Figure 3.1**, which needs to match the easting and northing parameters stated in the **Map Default Properties** dialog box (Figure 3.7, page 3-8).

```
Image West=2051443.
Image North=355443.           These are image boundaries
Image East=2054925.6          that are determined by the
Image South=351947.4          user and usually found using
                              a program like ArcMap®.



Image URL  =  path and filename of raster image
              (i.e., maps/lh_2m.gif)
```
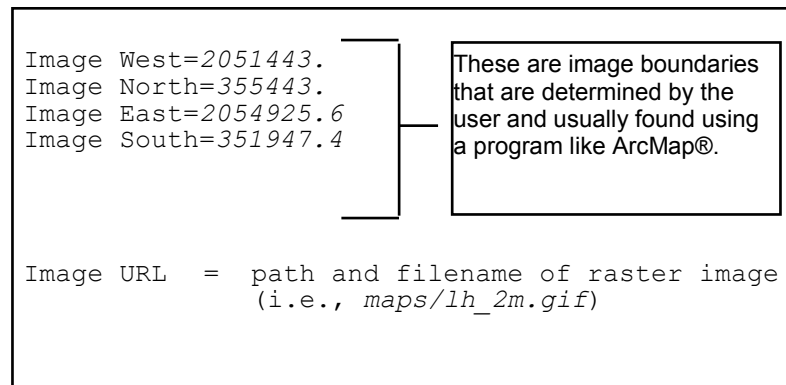
Figure 3.1 Extents of a raster image.

GeoTIFF (.tiff/.tif)

Users may also upload a GeoTIFF, which combines georeferencing information with the Tagged Image File Format. The information necessary to establish spatial reference will be contained within the file.

USGS Digital Line Graph (.dlg)

USGS Digital Line Graphs (DLG) are created from two sources using manual and automated digitizing methods. The two sources are aerial layer. Each shape has a record in the .dbf file containing descriptive data called "attributes." The .shx file contains an index that links each shape to its record in the .dbf file.

USGS DEM (.dem)

The USGS Geological Survey publishes the file format USGS DEM. A raster file format, *.dem* files usually contain terrain elevations, but the file format can be used for representing any surface. The format, published by the USGS, allows internal documentation of coordinate system, date of publication, etc. These values are referenced horizontally, either to a Universal Transverse Mecator (UTM) projection or to a geographic coordinate system.

ASCII NetTin (.net)

This is an ASCII file format used to represent a triangulated irregular network (TIN). TIN data are stored in a simple format that defines the nodes, edges, and triangles contained in the TIN. This format is common for transferring TIN data between software.

ArcInfo® DEM (.asc)

This is an ASCII file format representation of a grid. The file consists of keywords followed by values defining the number of columns, number of rows, lower left corner, cell size, no data value, and each cell value.

LizardTech® MrSID® (.sid)

The MrSID® (multiresolution seamless image database) is a propriety georeferenced image file format developed by LizardTech® for use with Geographic Information Systems (GIS). This format compresses large raster image files like aerial photographs or satellite imagery for easier viewing.

HEC-FIA SMRMP (.smrmp)

The Segmented Multi-Resolution Multi-Parameter (SMRMP) file was created to operate gridded data (namely, digital terrain grids, inundation depth grids, and arrival time grids) more efficiently in HEC-FIA. When users add terrain and inundation data (see Chapter 6), they will be prompted to convert to SMRMP format. The conversion is not required, but is recommended for higher efficiency.

3.2   Add Map Layers

Users can add as many map layers as needed. However, they must make sure that all map layers use the same measurement units and projection system. Map layers can be added either through the menu bar or the **Map Layers** folder of the **Study Pane**.

When a map layer is added to the study, FIA reads the coordinates of points, lines, and polygons on the map layer and resets the geographic extents of your watershed to the smallest rectangle that will contain all objects in the map layer. When adding more than one map layer, you must ensure that each is projected in the same coordinate system. For example, if you add a digital line graph (DLG) map layer that shows the major streams in an area, the geographical extents will change according to the projection of that DLG. If that DLG is projected in state plane

coordinates in feet, FIA will set the geographic extents accordingly, and the units of the stream alignment will be in feet.

To add a map layer:

1. Starting the **Study Pane**, right-click on the **Map Layers** subfolder. From the shortcut menu, select **Add Map Layers** (Figure 3.2).

2. The **Select Map Layer to Add** browser will open (Figure 3.3). Browse to the location where the required map layers are located.
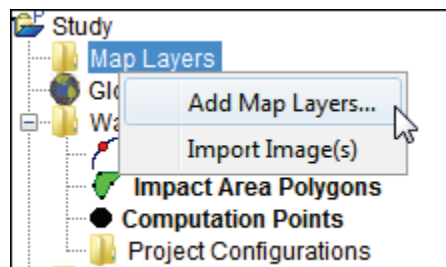
Figure 3.2 Adding map layers.

Select an individual map layer by clicking on the filename, or add multiple map layers by holding down the CTRL-key and selecting the map layers you would like to use for your study.
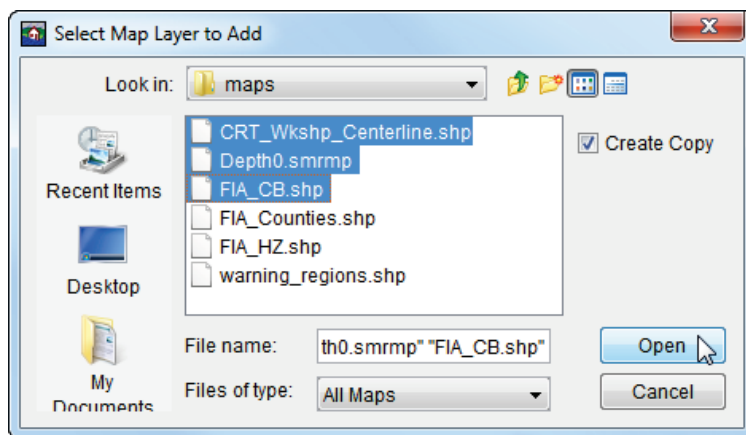
Figure 3.3 The Select Map Layer to Add browser.

3. **Create Copy** is selected by default; this will create a copy of the selected files in the *maps* folder within the current study directory. Storing all the data in the study directory is recommended as it will make the study easier to store and share.

4. Once the necessary files are selected, click **Open**. The **Select Map Layer to Add Browser** will close, and the map layers will display in the **Map Window**.

5. The filenames of the selected map layers will display under the **Map Layers** folder in the **Study Pane**.

6. To save your study and your newly-added map layer, from the toolbar click 🖫.

7. Users can also add map layers through the **Maps→Add Map Layers…** option. This will invoke the **Select Map Layer to Add** browser as the previous steps outlined.

3.3   Add Image

To add an image:

1.   In the **Study Pane**, right-click on the **Map Layers** subfolder. From the menu, select **Import Images** (Figure 3.2, page 3-5).
2.   The **Open** browser will open (Figure 3.4). Browse to the location where the images you wish to upload are located. Select an individual image by clicking on the filename, or add multiple images by holding down the CTRL-key and selecting the images you would like to use for your study.
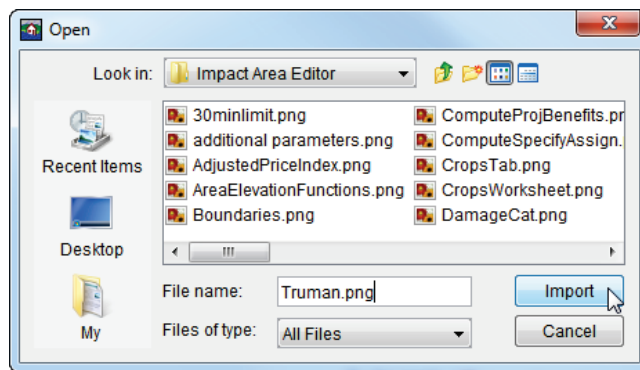
Figure 3.4 Open browser.

3.   Once the necessary files are selected, click **Import**. The **Open Browser** will close and the **Define Image Extents** dialog box will appear (Figure 3.5).
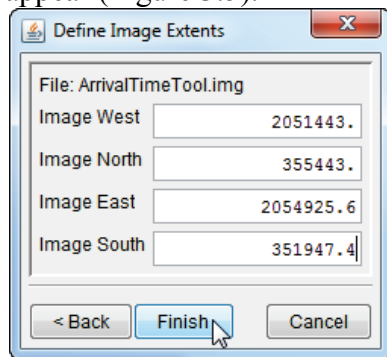
Figure 3.5 Define Image Extents dialog box.

4.   In the dialog, enter the extents of the image(s) you wish to upload. If you are uploading multiple images, fill in the extents of each image and click **Next >**. Once extents are defined for each image, the **Finish** button will replace the **Next >** button (Figure 3.5 above). Once you have entered extents for each image, click **Finish.**
5.   The filenames of the selected map layers will display under the **Map Layers** folder in the **Study Pane** and the image will be copied to the maps folder of the study directory.
6.   To save your study, from the toolbar click 💾 .

3.4   Remove Map Layers and Images

To remove map layers and images from your study:

1. Starting in the **Study Pane**, under the **Map Layers** folder right-click the name of the map/image you wish to delete.

Figure 3.6 Remove Map Layer.

2. From the shortcut menu, select **Remove Map Layer** (Figure 3.6).
3. The map layer or image will now be removed from your study.
4. The file for your map layer or image will remain on the hard drive. Should you wish to re-add a removed map layer or image, follow the instructions for adding a map layer or image outlined in Sections 3.2 and 3.3 above.
5. Should you wish to delete the map layer from your hard drive, open the **maps** folder of the study directory, right-click the file you wish to delete, and select **Delete**.

3.5   Geographic Reference for Map Layers

To maintain a geographic reference (also called a geo-reference), FIA uses a user-selected coordinate system, called the World Coordinate System (WCS). This superimposes a grid on feature layers to establish x- and y-coordinates in WCS for each point on the layer. The x-coordinate is referred to as "easting" and the y-coordinate is referred to as "northing". You can select the extent of this grid, the dimensions of the cells of the grid, the units of measurement of the grid, and even the location of the origin of the grid. Because of this flexibility, you can specify and use virtually any grid coordinates convenient for impact analysis.

To edit geo-reference settings for your study:

1. On the **Maps** menu, select **Default Map Properties.**
3. The **Map Default Properties** dialog box will appear (Figure 3.7). The information in the **Map Default Properties** dialog box includes:
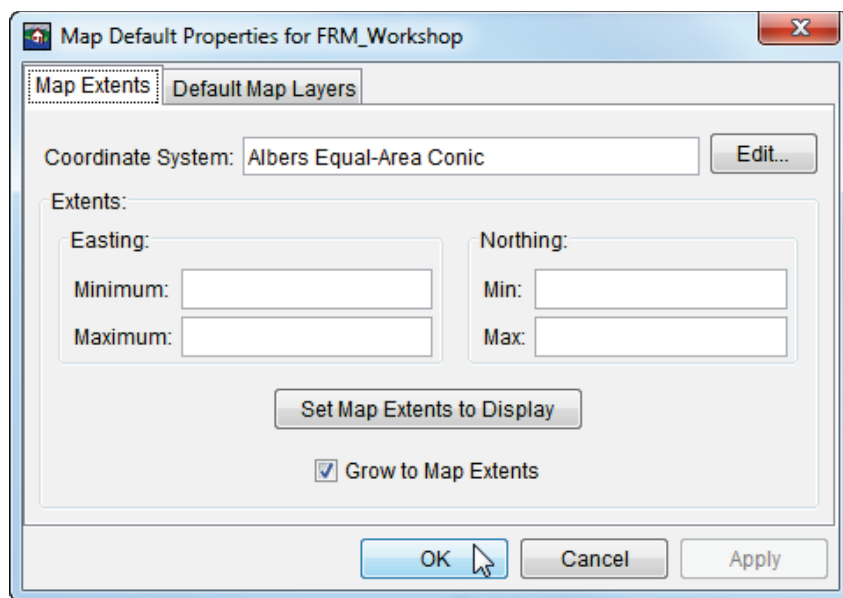
Figure 3.7 Map Default Properties dialog box.

- **Coordinate System.** This box identifies the established coordinate system for a study. By default, the Albers Equal-Area Conic coordinate system will be selected. To edit the coordinate system, click **Edit**, and the **Map Coordinate Information** dialog box (Figure 3.8) will open. Appendix D for details on editing the coordinate system. Remember that all layers must exist in one unified coordinate system.
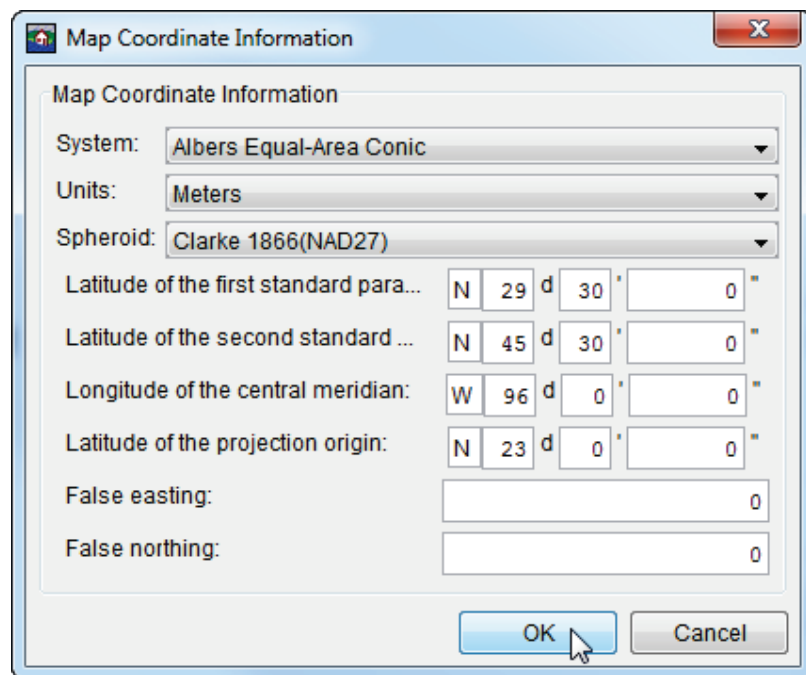
Figure 3.8 Map Coordinate Information dialog box.

- **Extents – Easting Minimum and Maximum, Northing Minimum and Maximum**. These values indicate the location of the left, right, bottom and top borders (respectively) of the grid in the display area.
- **Grow to Map Extents**. When this check box is selected, FIA automatically sets the geographic extents to define the smallest rectangle that encompasses all the objects in the watershed.
- **Set Map Display to Coordinates**. This will set the limits of the display area. If you zoom in on an area, and click **Set Map Display to Coordinates**, the extents on the **Map Default Properties** dialog box (Figure 3.7) will change to the zoomed area.

Note that the geographic extents of your layers must be selected carefully to ensure that the entire study is included. Further, the extents you specify and the coordinate system you use must be consistent for all the layers. Therefore, you may need to use GIS tools to transform the layers from one coordinate system to another before using them with FIA (Appendix B for GIS Pre-Processing information). FIA has no coordinate system transformation tools.

3.6   Map Tools

After opening a study, the following tools will be available on the toolbar in the map window:

Select Tool

Use the **Select Tool** to select different layer elements in the **Map Window**. Engage the tool and double click on a layer element (e.g. an Impact Area or structure in the Global Inventory) to select it. In Figure 3.9, the Impact Area Cowlitz has been selected.

Zoom Tool

Use the **Zoom Tool** to zoom in and out in the **Map Window**. Click and hold the left mouse button and draw a box in the map window to zoom in (Figure 3.10). Right click to zoom out.
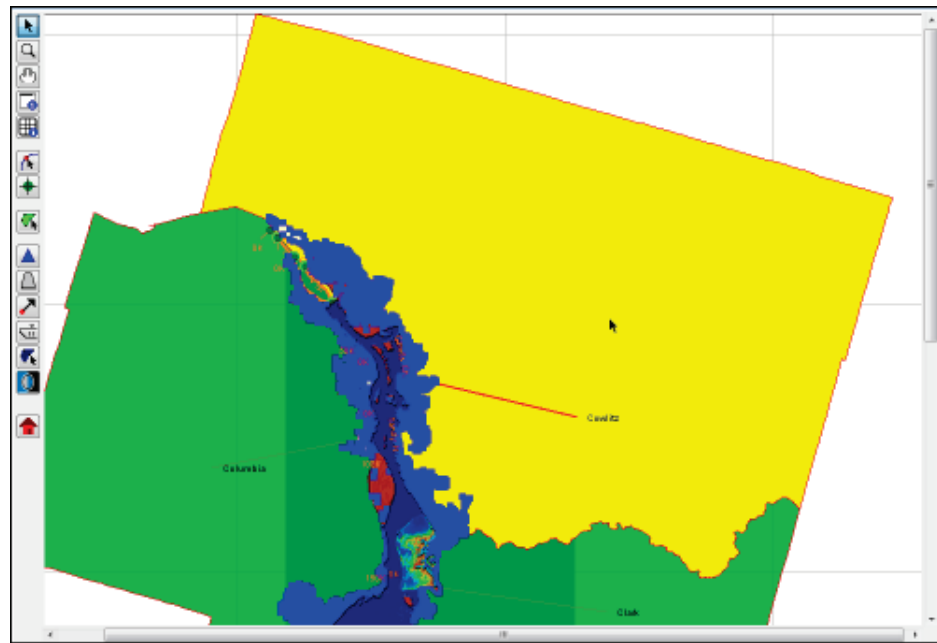
Figure 3.9 Selecting a layer element in the Map Window with the Select Tool.
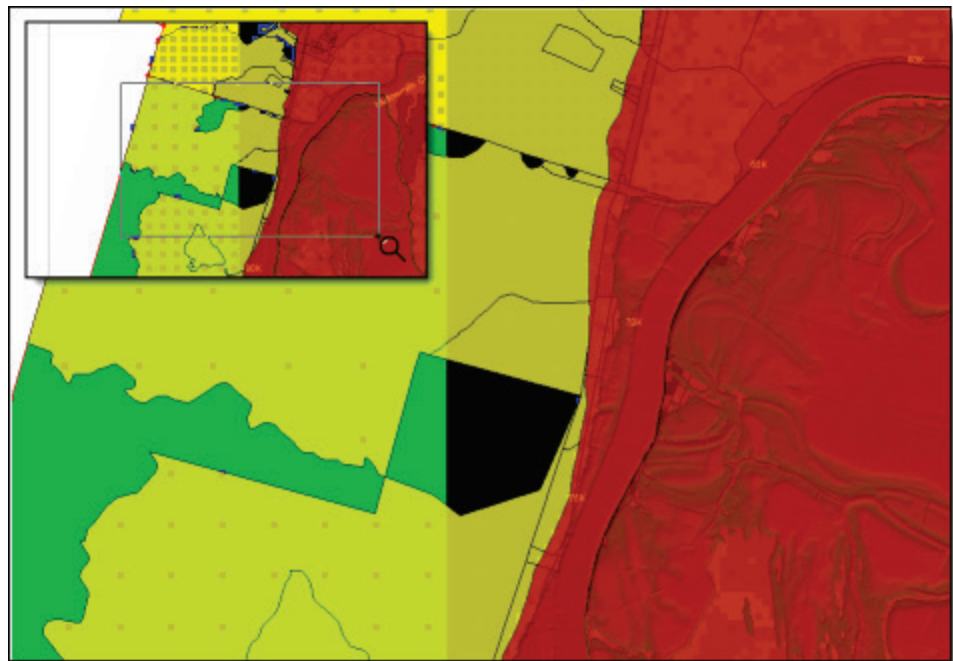
Figure 3.10 Using the Zoom Tool in the Map Window—Zoom Tool selection at upper left, with the result in the background.

Alternatively, users can zoom using the **scroll wheel** of their mouse while holding down the Ctrl-key.

Pan Tool

Use the **Pan Tool** to move on-screen display around without changing the scale. Click and hold the left mouse button to drag the display.

Shapefile Query Tool

Use the Shapefile Query Tool to view shapefile data for selected points in the **Map Window**.

To perform a shapefile query:

1. Starting in the **Map Window**, select the **Shapefile Query Tool** from the toolbar.

2. Using the cursor, click on the point on the map window at which you would like to perform a shapefile query.

3. The **Query Window** will appear. If you would like to query multiple locations of the map window, continue to click and the locations will add to the table in the **Query Window** (Figure 3.11).

OBJECTID	CensusBl...	Tract	BldgSche...	BlockType	BlockArea
147859.0	53011040...	53011040...	WA1	R	0.74
147848.0	53011040...	53011040...	WA1	R	5.905
208384.0	41051007...	41051007...	OR1	R	5.882
208826.0	41051007...	41051007...	OR1	R	7.8
201738.0	41051010...	41051010...	OR1	R	7.359
211130.0	41051010...	41051010...	OR1	R	2.397
142702.0	53015000...	53015000...	WA1	R	0.043
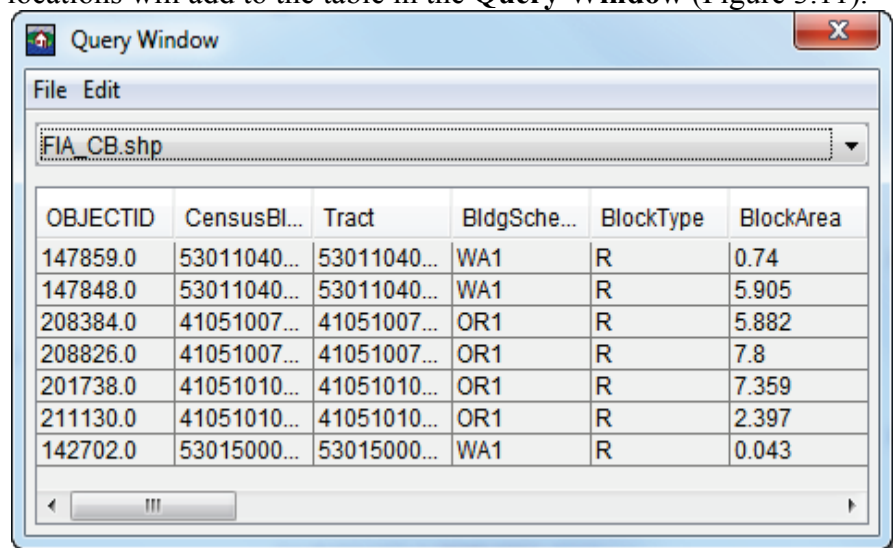
Figure 3.11 Query Window.

4. From the dropdown list, select the shapefile for which you would like to view query results. The data fields defined in the selected shapefile will display values for each point you selected with the query tool. In the example in Figure 3.11, the *FIA_CB* (Census Block) layer has been selected.

5. When finished viewing queries, close the window. The queries at the selected points may be viewed again using the query tool.

6. To clear queries, from the **Edit** menu, select **Remove Glyph** (or **Clear**).

3.7   Organizing Map Layers and Images

After map layers and images have been added, they can be turned off and on, moved in front of or behind other layers, or their viewing properties can be changed. To perform these operations on the map layers, from the FIA main window, click the **Maps Tab** just beneath the **Content Pane**. The **Content Pane** will now provide a tree of the available map layers, stream alignment, and information specific to what is being displayed by the selected map window (Figure 3.12).
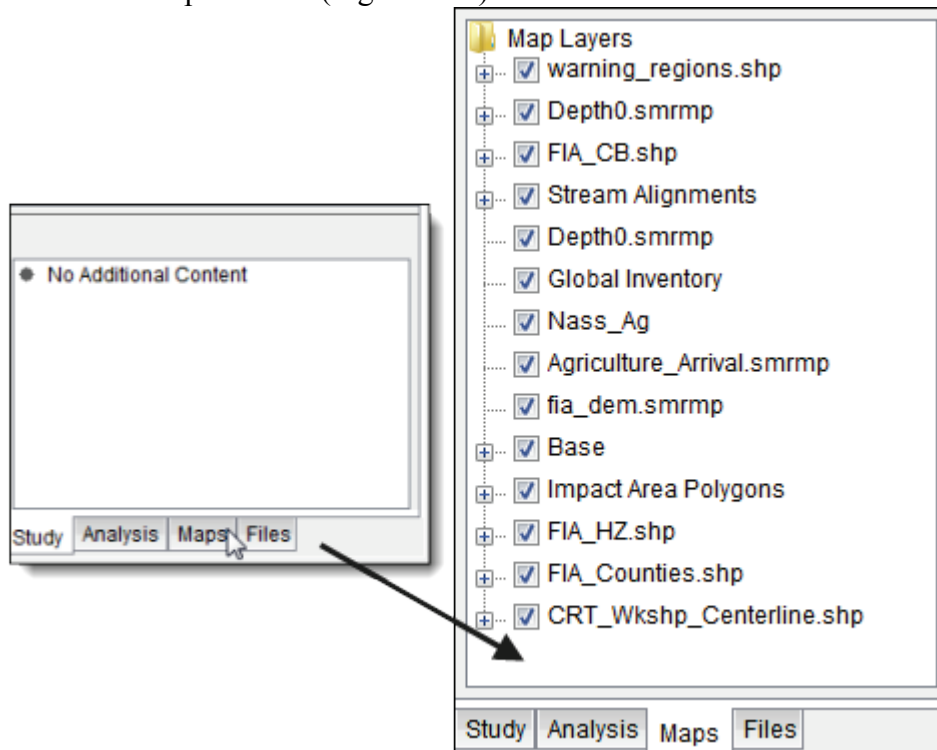
Figure 3.12 Click on the Maps tab to display layer information in the Content Pane.

3.7.1   Edit Display Order

To display or hide a map layer:

1. Starting in the **Content Pane** (Figure 3.12 above), locate the name of the map layer you wish to display/hide.
2. Check the box next to the map layer name to display it in the **Map Window**.
3. Clear the box to hide it from the **Map Window**.

Users may also toggle display order of study layers by moving them up or down:

1. Starting in the **Content Pane** (Figure 3.12), right-click the name of the map layer you wish to move.
2. From the shortcut menu select either **Move to Top**, **Move Up**, **Move Down**, or **Move to Bottom**.

3.7.2   Edit Display Properties

Users can also edit the way a layer displays in the **Map Window**, including the color of a line or polygon, labels, line styles and weight, level of transparency for polygons, and much more. The way these properties are edited are layer-specific and outlined in detail in Appendix E.
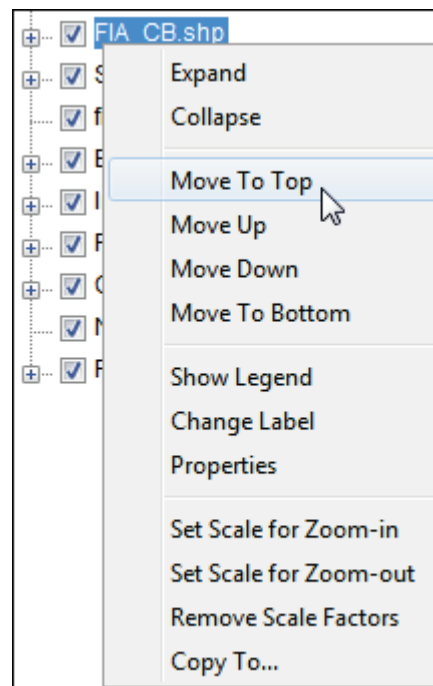
Figure 3.13 Map Layer shortcut menu (Content Pane).

To edit display properties of a map layer:

1. Starting in the **Content Pane** with the **Maps Tab** selected (**Figure 3.12**), select and right-click the name of the file you wish to edit.
2. From the shortcut menu, select **Properties** (Figure 3.13). A dialog box similar to **Figure 3.14** will appear.
3. Toggle display options as desired.
4. Click **Apply** to display your changes in the **Map Window** and keep editing.
6. When finished editing, click **OK** to close the dialog.
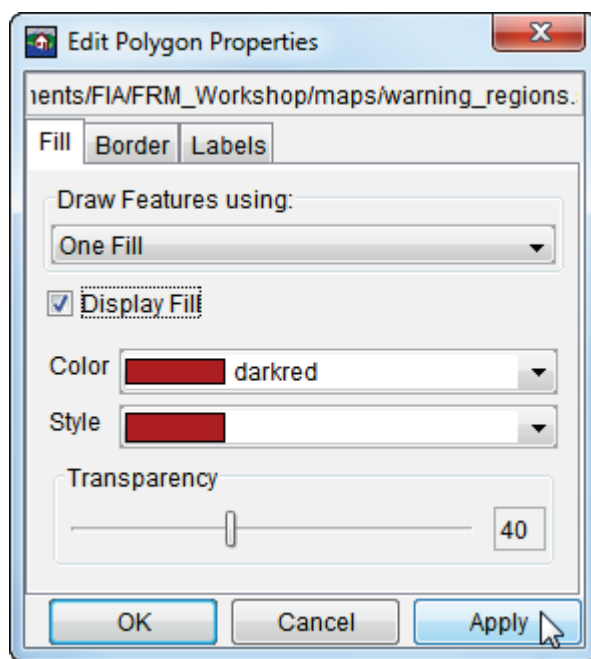
Figure 3.14 Edit Polygon Properties dialog box.

Chapter 4

Global Data

Global data is the information that describes a study in general terms. This chapter covers the concepts of global data and then details the information required for each tab in the **Global Data Editor**. To access the **Global Data Editor**, from the **Data** menu, click **Global Data**. Alternatively, in the **Study Pane**, right-click on **Global Data** and select **Edit**.

4.1   Concepts

HEC-FIA is designed to simplify entry and management of the large volume of data required for a flood impact analysis. The file system helps to reduce errors in data entry, promote consistency, eliminate redundancy in the data for a complex impact analysis, and simplify reporting tasks.

> *Global data develops a library of physical properties. These properties will be used later to define the individual impact areas.*
>
> *Another advantage is that the data is stored in an efficient fashion, without redundancy.*

FIA stores information that describes important characteristics, properties, or **attributes** of any item or object that is relevant to the impact analysis. For example, while you are entering the global data, you are also building the **global.fgd** file with the data that describes general information. In essence, you are developing a library of physical properties or identifiers that encompasses the entire study area. When users create **alternatives** from HEC-DSS hydrographs (see Chapter 9, Section 9.3, page 9-6), the information entered in the **Global Data Editor** will be used to define the individual impact areas.

Another advantage is that the data is stored in an efficient fashion, without redundancy. Suppose, for example, two impact areas both have the same crop (e.g., soybeans). Should FIA store the yield and price data twice, once for each impact area? Instead, FIA stores the yield, price, and other crop information in the **global.fgd** file and stores only the *reference* to the correct crop entry with the impact area information. It then refers to the crop information in the global data, looking up the correct yield and price to assess damage. This strategy simplifies what you must do to change the price; you need not change the entry for each impact area. Instead, you can change the entry in the global data only. The change will then propagate throughout the entire analysis.

The remainder of this chapter describes how to enter the data for the individual tabs - **Global Specs**, **Boundaries**, **Damage Categories**, and **Crops**.

4.2   Global Specs Tab

The **Global Specifications** tab, illustrated in Figure 4.1, contains unit and index factor information for a study.

> *The **Global Specs** tab contains unit and index factor information for a study.*

Figure 4.1 Global Data Editor—Global Specifications tab.

Definition of data entry items follows:

- **Monetary Units** – this value is used as a label for data entry forms and output reports. The choice does not influence the computations within FIA. From the **Monetary Units** list select from: *$1*; *$1000*; or *$100,000*. The default is *$1000*.
- **System Units** – this box displays the system units that were chosen when the study was created.
- **Urban or Agricultural Base Data Year** – this value is used as a label to specify the base year for both urban and agricultural data. This value along with the price index factor described below adjusts the past economic costs to current value. For example, if the economic data is from a structure inventory completed in 1985, enter 1985 in the **Urban Specifications** area in the **Base Data Year** box.
- **Urban or Agricultural Price Index Factor** – this multiplier is used to compute the current value of past economic costs, accounting for the time value of money. This applies to either urban or agricultural costs. The default is 1.0. FIA computes current costs from past costs as:

$$Current\ Value = Base\ Year\ Value * PIF$$

where:

> *PIF* = the price index factor. *PIF* can be computed using a number of methods. Two of the more common methods are described:

$$PIF = (1+i)^{CY-BY}$$

where:
> CY = the current year
> BY = the base year
> i = average discount rate over period, (e.g. 4% = 0.04), or,

$$PIF = BCI_C / BCI_B$$

where:
> BCI_C = Building Construction Index for the current year
> BCI_B = Building Construction Index for the base year.

To enter the global specifications:

1. By default the **Global Specs** tab should be displayed, if not click **Global Specs** tab (Figure 4.1).
2. From the **Monetary Units** list, select the appropriate monetary units for the study. In Figure 4.1, *$1000* has been selected.
3. In the **Urban Specifications** area, you may need to enter a base data year and price index factor. In the **Base Data Year** box, enter the year that represents the urban economic data. In Figure 4.1, *1995* has been entered for the **Base Data Year**. In the **Price Index Factor** box, enter a factor that will be used to update the base year economic data to current costs. In Figure 4.1, a **Price Index Factor** of *1.17* (assuming an average of four-percent inflation) has been entered.
4. In the **Agricultural Specifications** area, you may need to enter a base data year and price index factor. In the **Base Data Year** box, enter the year that represents the agricultural economic data. In Figure 4.1, *1999* has been entered for the **Base Data Year**. In the **Price Index Factor** box, enter a factor that will be used to update the base year economic data to current costs. In Figure 4.1, the default value has been used for the **Price Index Factor** since the agricultural economic data is current.
5. Click **Apply** if you are going to enter more global data. When finished entering data, click **OK**. The **Global Data Editor** will close.

The base year and price index factors that you entered in this tab are global for all impact areas in the workspace. When users create **Alternatives**, these factors can be overridden at the individual impact area. More information about specific impact area adjustments of global specifications is provided in Chapter 9, Section 9.7.8 (page 9-40).

4.3   Boundaries Tab

The **Boundaries** tab, illustrated in Figure 4.2, contains the global boundaries, which include physical, jurisdictional, and political boundaries. The types of boundary information that can be entered are: **Corps District**, **State**, **County**, **Township**, **Watershed**, **Subbasin**, **Congressional District**, **Community**, and **Flood District**. All boundaries are optional and are used for aggregation of damage/impact information for reports and displays.

The ***Boundaries*** *tab contains the global boundaries, which include physical, jurisdictional, and political boundaries.*
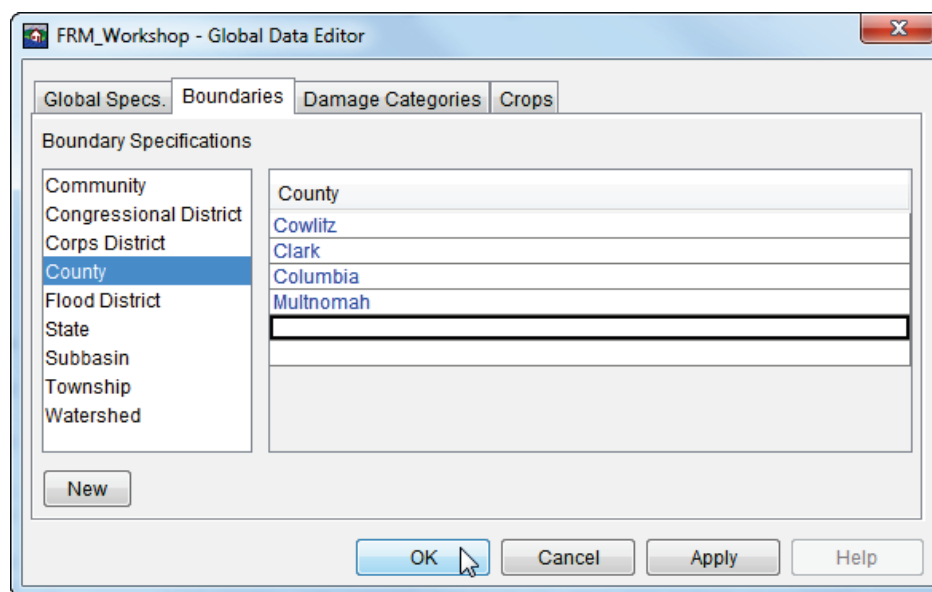
Figure 4.2 Global Data Editor—Boundaries tab.

To enter boundary data:

1. Click **Boundaries** tab, the **Boundaries** tab is now displayed (Figure 4.2).
2. From the **Boundary Specifications** list, select the boundary type you would like to define. In Figure 4.2, *County* has been selected and the column header of the table reflects that choice.
4. Once you have entered data, press **Enter**. The cursor will move to the next row. To add a row, right-click on the table, a shortcut menu will appear. If you are on the last row of the table with data entered, select **Append Row**. In Figure 4.2, *Cowlitz, Clark, Columbia,* and *Multnomah* have been entered as the counties for the study area.

5. Click **Apply** if you are going to enter more global data. When finished entering data, click **OK**. The **Global Data Editor** will close.

4.4   Damage Categories Tab

A **damage category** is a consolidation of structure types into specific categories with similar characteristics for analysis and reporting. In the **Damage Categories** tab you can define the damage categories for the study, rename damage categories, and delete damage categories. Examples of damage categories include residential, industrial, commercial, public, and undeveloped land. Later, if users compute damages from alternatives based on time-series hydrographs (as opposed to alternatives from gridded data; see Chapter 9, Section 9.2, page 9-5), they can input different damage functions for each damage category they create in the **Global Data Editor**. When users import structure inventories (see Chapter 8), four damage categories (Commercial, Public, Industrial, and Residential) will be created automatically. In a gridded compute, damage categories will not affect compute, but it may help the user organize their data for viewing results.

4.4.1   Add a Damage Category

*A **damage category** is a consolidation of structure types into specific categories with similar characteristics for analysis and reporting.*

You will provide the necessary information for the urban damage categories for a study. Definition of data entry items follows:

- **Damage Category** - a name that represents a damage category (required).
- **Reconstruction Period** - is the average number of days required to rebuild a damaged area to pre-flood economic conditions. Because some flood damage categories can recover faster than others, the reconstruction period is associated with a specific damage category.

  In time-series-based computes, FIA uses this value when computing damages caused by a multi-peak flood event. Damage from a second flood occurring during the reconstruction period is linearly proportioned to the depth of flooding in that zone and the number of days that has elapsed in the reconstruction period. For example, a damage category has a value of $100,000 for a particular zone and has a six-month reconstruction period. The zone is completely flooded and total loss occurs. Another flood of the same magnitude occurs four months after the first flood; the zone will experience $66,700 in damage, as it wasn't fully reconstructed when the second flood occurred. The reconstruction period cannot exceed 365 days (required).

To add a damage category:

1.   Click **Damage Categories** tab, the **Damage Categories** tab is now displayed (Figure 4.3).

<table>
<tr><td colspan="3">FRM_Workshop - Global Data Editor</td></tr>
<tr><td>Damage Category</td><td>Reconstruction Period (days)</td><td>Price Index</td></tr>
<tr><td>Commercial</td><td>180</td><td>1.00</td></tr>
<tr><td>Public</td><td>180</td><td>1.00</td></tr>
<tr><td>Industrial</td><td>180</td><td>1.00</td></tr>
<tr><td>Residential</td><td>180</td><td>1.00</td></tr>
</table>

*From the **Damage Categories** tab define the damage categories for the watershed.*
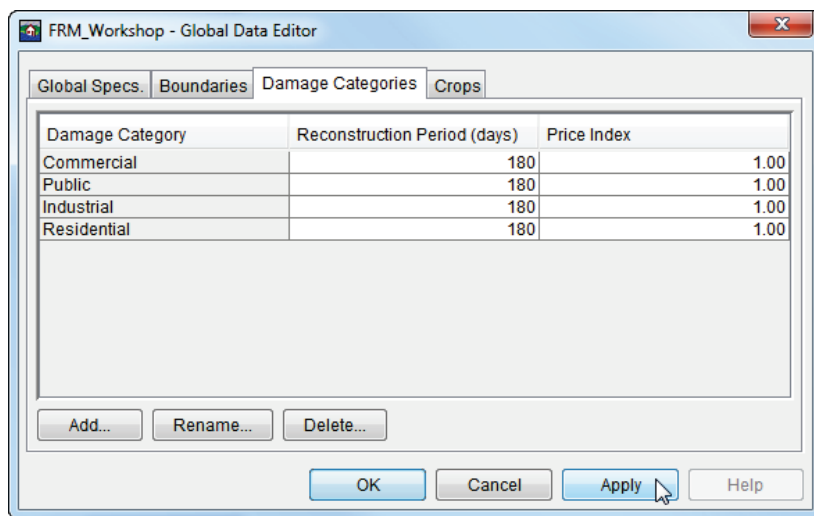
Figure 4.3 Global Data Editor—Damage Categories tab.

2.   Click **Add…** The **Add Damage Category** dialog box will open (Figure 4.4). In the **Damage Category** box, enter a name. In the **Reconstruction Period,** enter a number that reflects the reconstruction period. In Figure 4.4, *Undeveloped Open Space* has been entered for **Damage Category**, and *0* (zero) has been entered for the **Reconstruct ion Period**.

FRM_Workshop - Add Damage Category

Damage Category: Undeveloped Open Space

Reconstruction Period: 0
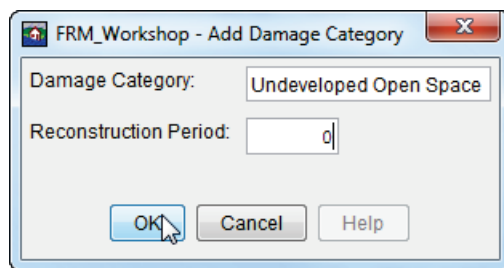
Figure 4.4 Add Damage Category dialog box.

3.   Click **OK**. The **Add Damage Category** dialog box will close and the entered damage category and reconstruction period will appear on the **Damage Categories** tab in a table.

4.   Repeat steps 4 and 5 for each damage category to be entered.

5.   Click **Apply** if you are going to enter more global data. When finished entering data, click **OK**. The **Global Data Editor** will close.

From the **Damage Categories** tab (see Figure 4.3, page 4-6), users can also rename or delete damage categories.

4.4.2   Rename a Damage Category

To rename a damage category:

1. Open the **Global Data Editor** and select the **Damage Categories** tab (Figure 4.3).
2. At the bottom of the editor, click the **Rename…** button. The **Rename Damage Category** dialog box (Figure 4.5) will open. In the **From** list, select the damage category you wish to rename. In the **To** box, enter the new name for the damage category.
3. Click **OK**. The **Rename Damage Category** dialog box will close and the new name will appear in the table on the **Damage Categories** tab (Figure 4.3).
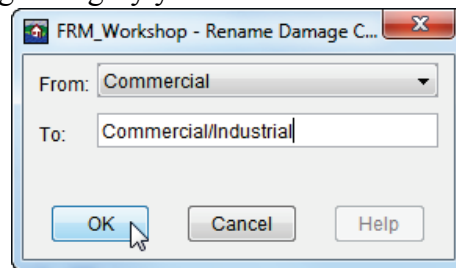
Figure 4.5 Rename Damage Category dialog box.

4.4.3   Delete a Damage Category

To delete a damage category:

1. Open the **Global Data Editor** and select the **Damage Categories** tab (Figure 4.3).
2. At the bottom of the editor, click the **Delete** button. The **Delete Damage Category** dialog box (Figure 4.6) will open. A list of available damage categories appears; select the damage category to be deleted (it will be highlighted in blue). In Figure 4.6, *Residential* has been selected.
3. Once the correct category is selected, click **OK**. The damage category will be deleted.
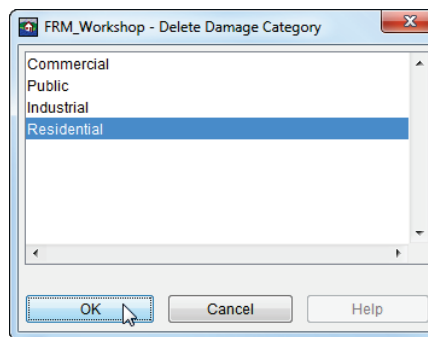
Figure 4.6 Delete Damage Category dialog box.

4.5   Crops Tab

If you are going to be computing agricultural damages from HEC-DSS time-series hydrographs, agricultural information for the workspace is entered in the **Crops** tab. FIA considers yield and value of crop; timing and duration of flooding; effects of replanting and late planting; and time

between flood events when computing flood damages in agricultural areas. Monetary damage values are determined from investment losses, mature-crop price values, and harvest costs. The **Crops** tab (Figure 4.7) is where you will enter a comprehensive list of all crops in the study area, along with their respective flood-damage potential.

*If users are computing damages using time-series hydrographs, a comprehensive list of all crops in a study area must be entered in the **Crops** tab. This function is not relevant to a grid-based compute.*

Figure 4.7 Global Data Editor—Crops tab.

If, on the other hand, you will be computing agricultural damages based on gridded data (like a NASS cropland data layer grid), you need not enter information in the **Crops** tab. Instead, this information will be contained and defined within the uploaded agricultural grid. See Chapter 7, Section 7.5 (page 7-7) for defining crop loss information using gridded data.

4.5.1   Add a Crop

To add a crop:

1. Open the **Global Data Editor** and select the **Crops** tab (Figure 4.7).
4. Click the **Add…** button. The **New Crop Name** dialog box (Figure 4.8) will open. In the **New Crop Name** box, enter a name.

Figure 4.8 New Crop Name dialog box.

5. Click **OK**. The dialog box will close and the name of the crop will appear on the **Crops Tab** in the **Name** list (Figure 4.7). Once a new crop has been entered, information needs to be entered which defines the crop.

4.5.2   Crop Value Information

Crop value information is used to develop the maximum crop value (also equal to the investment plus profit) at harvest. The information required includes:

> *Crop value information is used to develop the maximum crop value (also equal to the investment plus profit) at harvest.*

- **Yield Per Acre** - this is the average number of units produced per acre of land on which the crop is planted when the crop is harvested.
- **Unit** – is used to measure the amount of harvested crop. For example, corn may be measured in bushels.
- **Unit Price Value** – value in dollars of a single unit of harvested crop.
- **Harvest Cost/Acre** - the amount in dollars it costs to harvest one acre of the crop.
- **Additional Losses % Total** –this loss is estimated as the percentage of the mature crop value that will be added to the total damage to account for damage to farm equipment, roads, etc., during inundation of the crop.

To enter crop value information:

1. Open the **Global Data Editor** and select the **Crops** tab (Figure 4.7).
2. In the **Description** box, users may enter a description of the crop (Figure 4.9).
3. In the **Yield Per Acre** box, enter a number that represents the average number of units produced per acre of land.

Description:	
Yield Per Acre:	30
Unit:	Ton ▼
Unit Price Value $:	0.18
Harvest Cost/Acre $:	80
Addnl Losses % Total:	0

Figure 4.9 Detail—Crops tab.

4. In the **Unit** box, enter or select the units that measure the amount of the harvested crop.
5. In the **Units Price Value** box, enter the value in dollars of a single unit of the harvested crop.
6. In the **Harvest Cost/Acre** box, enter the amount in dollars it cost to harvest one acre of the crop.

7. In the **Addnl Losses % Total** box, enter a percentage which will be added to the total agricultural damage to account for farm equipment, roads, etc., during inundation.

8. Click **Apply**. Once you have the crop value information entered, you next need to enter the required information for defining the crop loss functions.

4.5.3 Double Cropping

*A double crop (secondary crop) can be planted in the same field during the same year after the harvest of the first crop (primary crop) this is **double cropping**.*

Sometimes, a second crop (secondary crop) can be planted in the same field during the same year after the harvest of the first crop (primary crop). This is called **double cropping**. FIA allows you one or more double crops for an impact area. Individual crop loss functions will be defined for both crops; however, conceptually the crop functions of the two crops would be combined into one crop function (Figure 4.10). The damage and benefits of both crops will be computed as well as the area flooded for both crops.
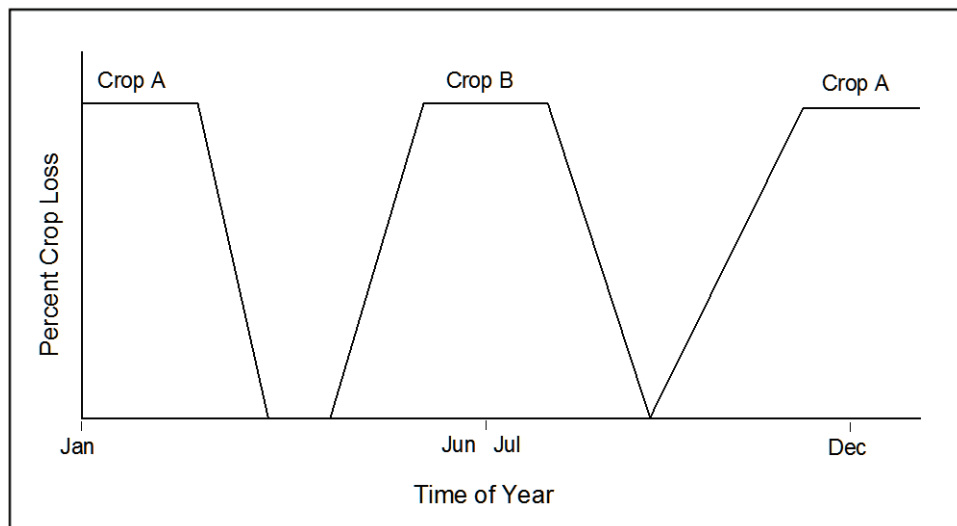
Figure 4.10 Double Crop-loss Function.

To designate a crop loss secondary:

1. Open the **Global Data Editor** and select the **Crops** tab (see Figure 4.7, page 4-8).

2. By default a crop is designated a **Primary Crop**. In Figure 4.11, *Corn* is designated as a primary crop. If you want to designate the crop as a secondary crop, simply select **Secondary Crop**.

Figure 4.11 Crop designation (Global Data Editor—Crops tab).

4.5.4   Define Crop Loss Functions

> *A **crop loss function** defines the crop damage potential for a specific crop as it varies throughout the year from soil preparation to completion of harvest.*

The following is relevant only to studies with damage computation in the form of time-series DSS hydrographs. *For crop loss definition with gridded agricultural data, see Chapter 7, Section 7.5.1.*

A **crop loss function** defines the crop damage potential for a specific crop as it varies throughout the year from soil preparation to completion of harvest. FIA uses two crop loss functions, one is **Initial Planting**, and the other is **Late Planting**.

The **Initial Plant Loss Function** represents the damage incurred on any given date if the crop did not have to be replanted due to flooding, and if planting was not delayed due to previous flooding. The **Last Plant Loss Function** represents the damage incurred, given the crop would not reach its full-yield potential because of late planting or re-planting. The functions depict a continuous relationship of day-of-year versus the percentage of the mature crop value that is lost, as illustrated by Figure 4.12.

When computing agricultural damage due to a flood defined by a flow or stage time series, FIA assumes that all crops are planted on the **First Plant Date**, unless the impact area is inundated or in the **Dryout Period** from a previous event. In that case, the program assumes the crop is planted as soon as the **Dryout Period** has ended. If a crop is planted, and a flood occurs, FIA assumes the crop is replanted if the soil dries before the **Last Planting Date**.
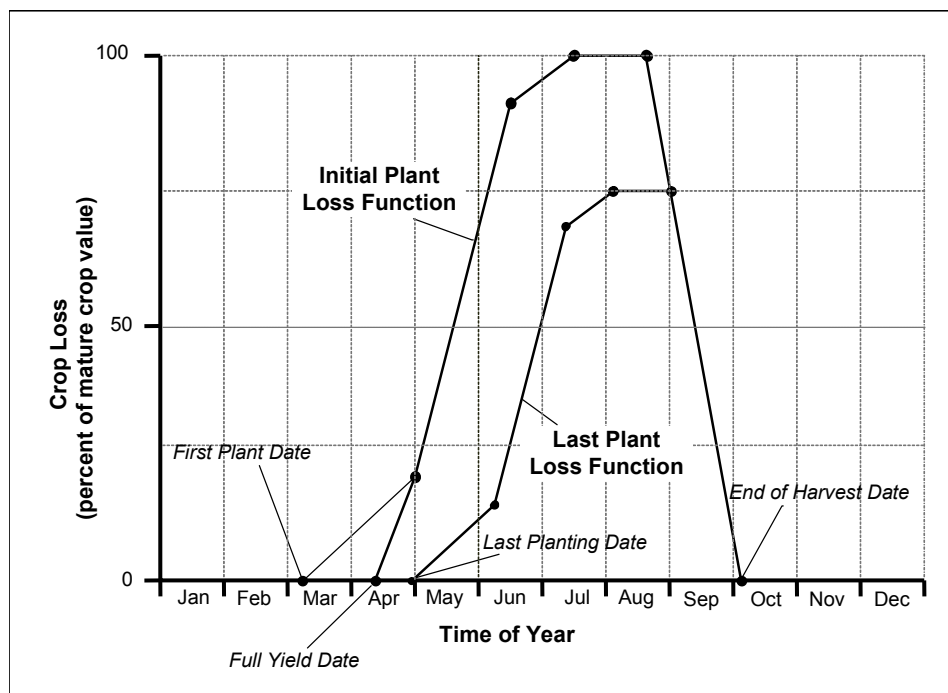
Figure 4.12 Initial and Last Plant Crop Loss Functions.

The value of a crop at maturity is computed as:

$$V = (Y * UP) - HC$$

> The ***Initial Plant Loss Function*** *represents the damage incurred on any given date if the crop did not have to be replanted due to flooding, and if planting was not delayed due to previous flooding.*

where:
V = the value per acre
Y = the yield per acre
UP = the unit price
HC = the harvest cost per acre

The total damage to a crop reported by FIA is equal to the percent damage from the crop loss function, multiplied by the mature crop value, plus the additional losses, which are also a percentage of the mature crop value.

For example, Table 4.1 shows potential damage values for corn. Using the initial plant loss function in Figure 4.12, if a field of corn was flooded on August 1st, the damage reported would be (1.00 * $236) + (0.10 * $236), for a total of $259.60/acre. However, if instead that field of corn was flooded on June 1st, the damage reported would be (0.65 * $236) + (0.10 * $236) for a total of $177/acre. The percent loss value of 0.65 is estimated from the initial plant loss function in Figure 4.12 for June 1st. The value of 0.10 accounts for additional losses and is taken from Table 12.1.

Table 4.1 Example Crop Loss Data for Corn

Crop	Yield/ Acre	Unit Price Value	Gross Value/ Acre	Harvest Cost/ Acre	Mature Value/ Acre	Additional Losses% Total
Corn	112	3.00	$336	$100	$236	10
Oats	50	2.00	$100	$35	$65	10
Soybeans	43	6.50	$280	$100	$180	15
Wheat	45	4.00	$180	$80	$100	10

If planting is delayed until the **Last Planting Date** due to a previous flood event, the same method is used to compute damages. However, FIA will pick the percent loss value from the **Last Plant Loss Function** instead of the **Initial Plant Loss Function**. Furthermore, if planting occurs between the **Full Yield Date** and the **Last Planting Date,** FIA will linearly interpolate a crop loss function between the **Initial Plant Loss Function** and **Last Plant Loss Function**, depending on the date the soil dries.

Seasonal Crop Data

Seasonal data and the actual values of the crop loss function define a crop loss function. Seasonal crop data includes: first day of planting, last day of planting which will produce a full yield, last day of planting which will produce a partial yield, and the last day of harvest. The entered calendar

days are converted to Julian calendar days by the program for calculation purposes.

- **First Plant Date** – first date that the crop can be planted.
- **Full Yield Date** – latest date that the crop can be planted and still reach full maturity.
- **Last Planting Date** – latest date that the crop can be planted with a reduction in yield.
- **End of Harvest Date** – last date that the crop is harvested, after which little damage can be incurred.
- **Dryout Period (days)** – number of days after a flood has receded before the soil is sufficiently dry so replanting can begin. FIA automatically adds seven (7) days on to the dryout period to allow for re-cultivation of the fields.

To enter seasonal crop data:

1. Open the **Global Data Editor** and select the **Crops** tab (see Figure 4.7, page 4-8).

2. In the **First Plant Date** box, type the first date a crop can be planted. The format is ddmmm. For corn (Figure 4.13) that is *07May*.

3. For the **Full Yield Date** box, enter the latest date that a crop can be planted and still reach full maturity. The format is ddmmm. In Figure 4.13, *01Jun* is the date entered for corn.

First Plant Date:	07May
Full Yield Date:	01Jun
Last Planting Date:	01Jul
End of Harvest Date:	15Nov
Dryout Period (days):	14

Figure 4.13 Seasonal Crop Data (Global Data Editor—Crops tab).

4. In the **Last Planting Date** box, enter the latest date that a crop can be planted with a reduction in yield. The format is ddmmm. For the example in Figure 4.13, *01Jul* has been entered for corn.

5. In the **End of Harvest Date** box, enter the last date that a crop can be harvested where little damage can occur. The format is ddmmm. For corn (Figure 4.13) that is *15Nov*.

6. In the **Dryout Period** box, enter a number that represents the number of days after a flood when replanting can begin. Figure 4.13, a value of *14* has been entered for corn.

7. Click **Apply**. Once you have the seasonal crop data entered, you next need to define the crop loss functions.

The data required for the crop loss functions are:

- **Date** – calendar days that are significant on the initial or last plant loss functions. Use the format *ddmmm*. For example, use *15OCT* for October 15 or *02May* for May 2.

- **Initial Plant Loss %** - the percentage of the mature crop value that is reported as damage, if the crop was planted on the **First Plant Date**.
- **Last Plant Loss %** - the percentage of the mature crop value that is reported as damage, if the crop was planted on the **Last Planting Date**.

To enter the values for the crop loss functions:

1. Open the **Global Data Editor** and select the **Crops** tab (see Figure 4.7, page 4-8).
2. Click **Define Crop Loss Function**. The **Crop Loss** dialog box (see Figure 4.14, page 4-14) will open. The name of the crop for which you are defining the crop loss functions will appear on the title bar.
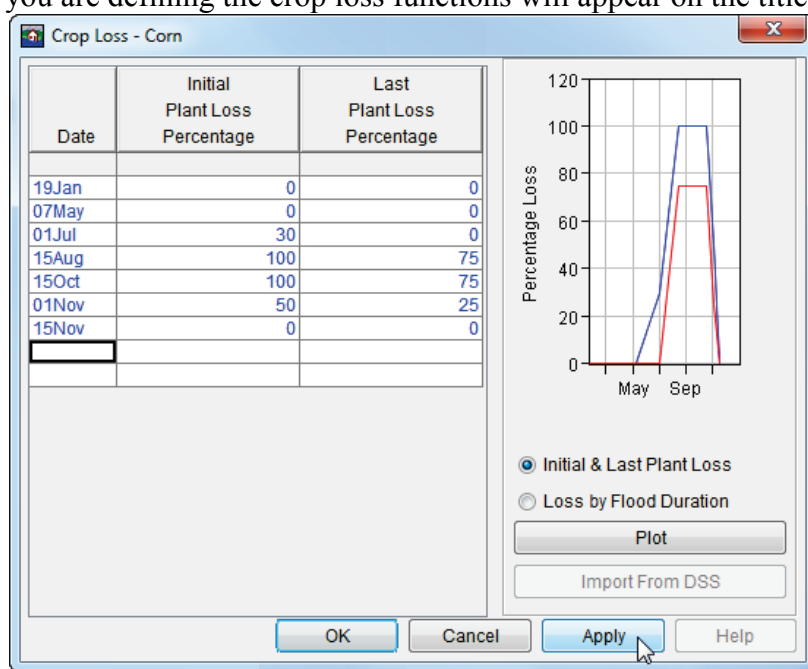
Figure 4.14 Crop Loss dialog box.

3. Here you will enter the data for crop loss functions. The data entry is done in a tabular format, with the **Date** column being the same for both functions.
4. In the **Date** column of the table, enter points during the year that have significance for the crop loss function. These points should include the **Seasonal Crop Data** items that have already been entered.
5. The values entered in the **Initial Plant Loss** column are percentages, which correspond to the dates entered in the **Date** column. These values represent the percent loss associated with initial planting – no reduction in yields from replanting or late planting resulting from a previous flood event. Intermediate values are linearly interpolated.

6. The values entered in the **Last Plant Loss** column are percentages, which correspond to the dates entered in the **Date** column. These values represent a reduction in yield and define the last plant loss crop function that could be brought into production after a flood event. Intermediate values are linearly interpolated.

7. To add a row, right-click on the table. A shortcut menu will appear. If you are on the last row of the table with data entered, select **Append Row**.

8. As values are entered into the table, the hot plot to the right of the table on the **Crop Loss** dialog box will continually update. To view a full-size plot, click **Plot**.

9. When finished entering crop loss function data, click **OK,** which closes the **Crop Loss** dialog box and returns you to the **Crops** tab.

4.5.5   Define Loss by Flood Duration Function

*Loss by Flood Duration functions adjust the crop damage per acre computed with the **Initial and Last Plant Loss functions,** based on the length of time the crop is inundated and the time of year the flood event occurs.*

Loss by Flood Duration functions (Figure 4.15) adjust the crop damage per acre computed with the **Initial and Last Plant Loss** functions, based on the length of time the crop is inundated and the time of year the flood event occurs. These functions account for the impact of the duration of inundation on crop damage. For example, in some cases, a crop that is inundated for one day is minimally affected, whereas a crop that is inundated for fourteen days could be destroyed. A maximum of six duration relationships may be entered. Careful definition of **the Loss by Flood Duration** functions is important because intermediate values are linearly interpolated.

Figure 4.15 Loss by Flood Duration Functions.

1. Open the **Global Data Editor** and select the **Crops** tab (see Figure 4.7, page 4-8).
2. Click **Define Crop Loss Function**, the **Crop Loss** dialog box will open (Figure 4.14). The name of the crop for which you are defining the crop loss functions will appear on the title bar.
3. Click **Loss by Flood Duration**, the **Crop Loss** dialog box will change (Figure 4.16).
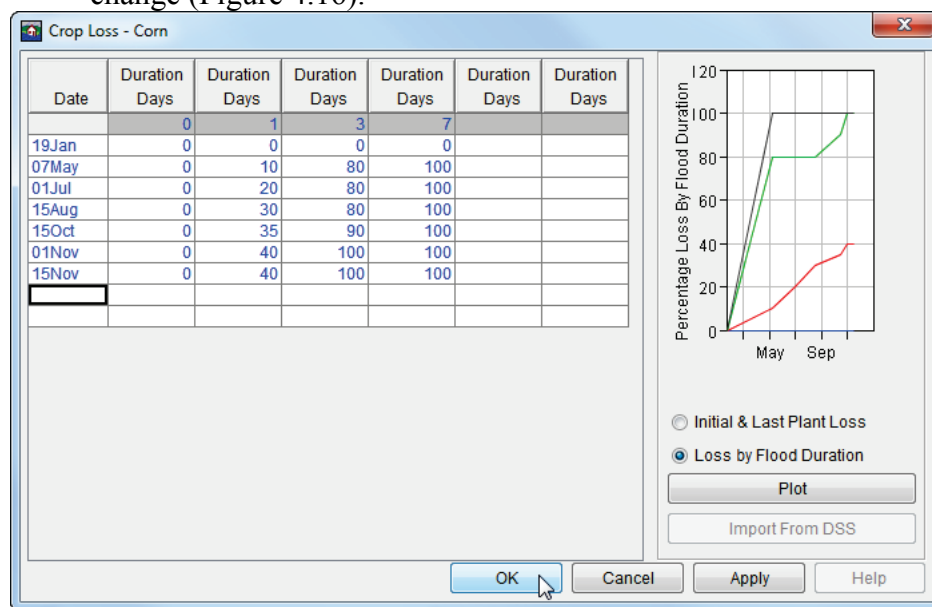
Figure 4.16 Crop Loss by Flood Duration.

4. The **Date** column will be automatically filled in with the data that was entered for the **Initial and Last Plant Loss** functions.
5. In the gray row, enter a set of duration values to define all loss-duration functions for a crop. For example, for the functions shown in Figure 4.16, you would enter *1* for flood duration of one day, and *7* for flood duration of seven days. The defined duration should be sufficiently long enough so that one hundred percent damage is incurred. Each column in the table represents a **Loss by Flood Duration** function.
6. The values entered in each **Duration** column correspond to the estimated percent of crop that would be lost for that particular duration of flooding. In Figure 4.16, 50% is entered as *50*.
9. To add a row, right-click on the table, a shortcut menu will appear. If you are on the last row of the table with data entered, select **Append Row**.
8. As values are entered into the table, the hot plot to the right of the table on the **Crop Loss** dialog box will continually update. To view a full-size plot, click **Plot**.
9. Click **OK,** which closes the **Crop Loss** dialog box and returns you to the **Crops** tab.

4.5.6   Delete a Crop

To delete a crop:

1. Open the **Global Data Editor** and select the **Crops** tab (see Figure 4.7, page 4-8).
2. From the **Name** dropdown list, select the crop you would like to delete.
3. Click **Delete**. A warning message will open asking you whether you really want to delete the selected crop. Click **Yes**. The warning message will close and the crop will be deleted from the **Name** list.

Chapter 5

Watershed Setup

A *common schematic* is used to effectively share common data between HEC software (i.e., ResSim) for a watershed. The common schematic consists of a stream alignment, map layers (Chapter 3), time series icons, computation points, impact areas, and projects. Another element included in the common schematic is a configuration. Configurations are a physical arrangement of the projects defined for a watershed that will be modeled by HEC software. This reduces the possibility of incompatible data sets from HEC software being used together.

From the **Map Window**, you will use tools to create the common schematic. In Chapter 2, Section 2.1.3 (pages 2-6 through 2-9), you are given an overview of the **Map Window** and the available tools. These tools allow you to create a stream alignment, computation points, impact areas, projects, and configurations.

5.1   Establishing the Stream Alignment

The first item to create in the common schematic is the stream alignment. A stream alignment is the backbone for all of the other elements that make up a common schematic. This section will describe the tools and concepts for creating a stream alignment.

5.1.1   Definition of a Stream Alignment

Stream alignments represent the stream system in the watershed. The alignment indicates where confluences and bifurcations occur and provide a sense of distance and scale. In FIA, stream alignments are composed of a series of stream elements, stream nodes, and stream junctions. A **stream element** typically begins or ends at a confluence, bifurcation, or at the boundary of the river system. A stream element is composed of vertex points, with the beginning and ending vertex points being stream nodes.

Stream elements are drawn as multi-segmented lines. Each stream element must have at least two stream nodes, defining the beginning and end of the stream element. Stream elements, stream nodes, and stream junctions, are illustrated in Figure 5.1.
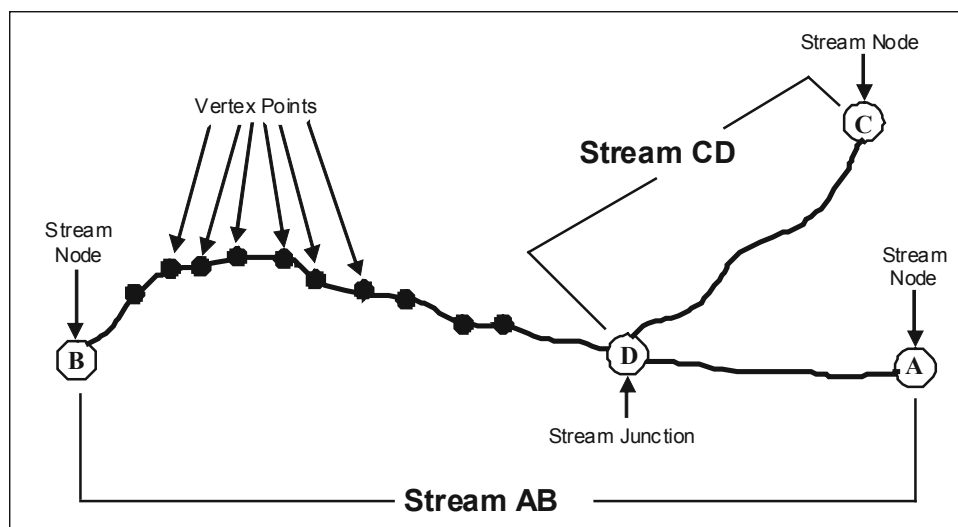
Figure 5.1 Relationship of Stream Elements in a Stream Alignment.

FIA creates and labels stream stations along each stream element. The ending stream node (the last point selected) of a stream element will always be labeled as Station 0. Therefore, if you want Station 0 to be at the downstream end of the stream element, begin drawing the stream element from the upstream end.

Stream stationing is based on the geographic extents that are defined when a stream element is drawn. If the geo-extents are based on a map layer, the units for the stream stationing are the same as the map layer (i.e., feet, meters). In Version 2.1 of FIA there is no indication of units. Keep in mind that map projection can determine units. For example, if a study is in Albers Equal Area Conic projection, the units will be in meters.

A stream alignment created without an underlying map layer will use the default coordinate system of X-Y. This default does not have any units attached to it. More information on the coordinate system is contained in Appendix D.

5.1.2   Create a New Stream Alignment

If your stream alignment is going to be based on a map layer, the map layer should already have been included in the watershed. The details on how to include a map layer are given in Chapter 3 (see Section 3.2, page 3-4). The map layer should contain water features, such as stream channels and reservoirs. Use this map layer as a guide while creating the stream alignment. By doing this, the units of the labels on the stream alignment will be the same as the units of the water feature layer (i.e., if your water feature layer is projected in state plane coordinates (feet), then the labels on the stream alignment will be in feet).

You can either digitize a stream alignment by hand or import a stream alignment from a shapefile. The following are the steps for digitizing a stream alignment. See Section 5.1.9 for details on importing a stream alignment.
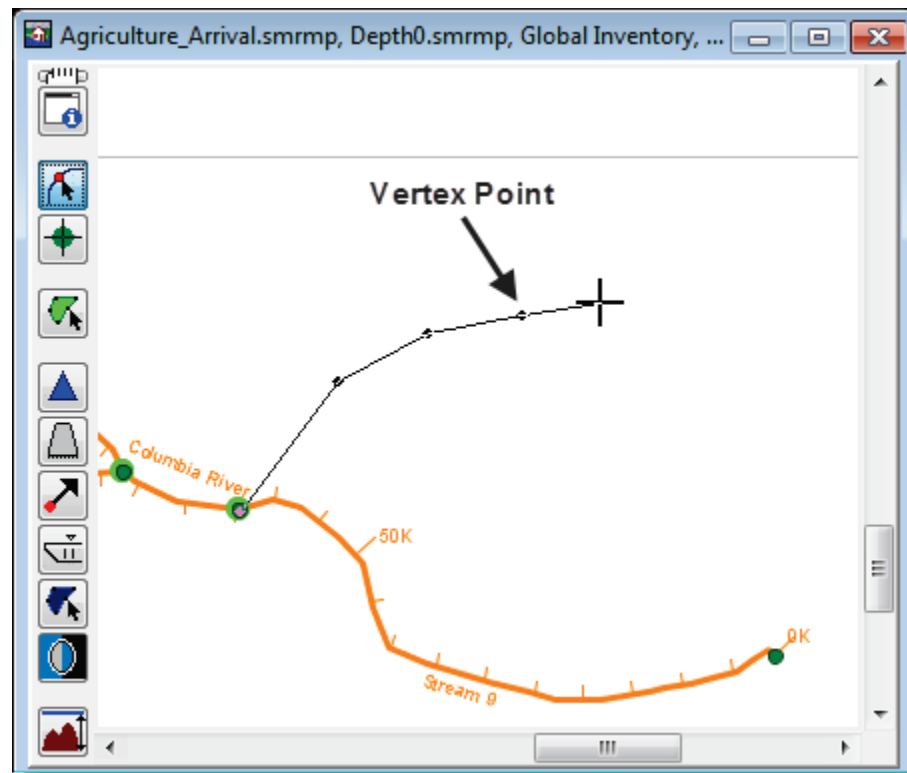
Figure 5.2 Creating a Stream Element.

Stream
Alignment
Tool

To create a stream alignment:

1. You must be in the **Map Window**.
2. On the map toolbar, press the **Stream Alignment Tool**.
3. Find a location on the **Map Window** where you want to start a stream element. Stream elements should be created from upstream to downstream. Hold down the **CTRL** key and click the mouse button. Each click creates a vertex point on your stream element (Figure 5.2). The first vertex point is the upstream or beginning stream node of a stream element.
4. For the most downstream or last vertex point on your stream element, release the **CTRL** key, and then click the mouse button. This creates the ending stream node of the stream element.
5. The **Create New Stream** dialog box will open (Figure 5.3). You can either accept the default name or enter a new name. You can also enter an optional description. Click **OK**. The **Create New Stream** dialog box will close.
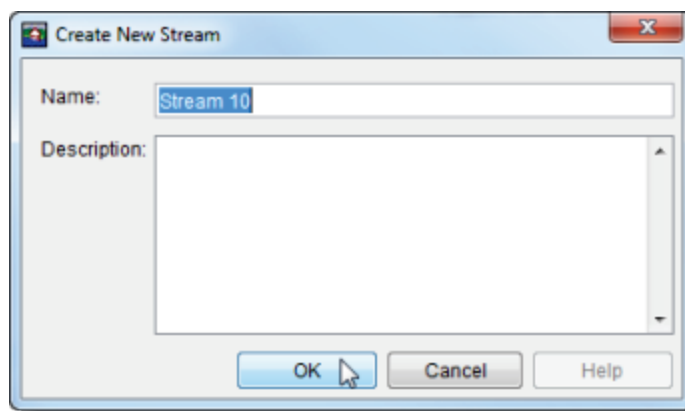
Figure 5.3 Create New Stream dialog box.

6. Repeat steps 3 through 5 to add more stream elements to your stream alignment.

7. To connect a new stream element to an existing stream element, place the first or last click of the new stream element on the existing stream element where you want the two stream elements to join. After naming the new stream element, a message box will appear asking if you want to connect the new stream element to the existing stream element. Click **Yes** if you want the two stream elements connected. Otherwise, click **No,** and the new stream element will not be connected to the existing stream element.

8. To save your study and your newly-added stream element, from the toolbar click 🖫.

The **Zoom Tool** (see Chapter 3, Section 3.6, page 3-10) can be used to help define the stream alignment. Use it to magnify the stream network, which makes defining the vertices along the stream easier.

5.1.3   Edit an Existing Stream Alignment

After drawing a stream alignment in the **Map Window**, you can rename and edit the description of an individual stream element, delete individual stream elements, reverse the direction of a stream element, disconnect a stream element at either the upstream or downstream end, and view the stream stationing and coordinates of stream nodes on an individual stream element. All of these tasks are
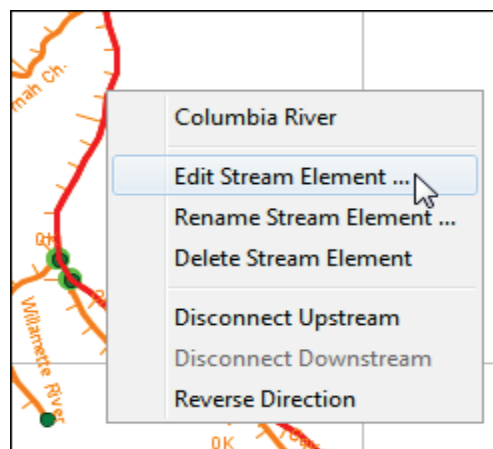
Figure 5.4 Stream Element shortcut menu.

accomplished through the stream element shortcut menu (Figure 5.4). To access the stream element shortcut menu, you must be in the **Map Window** and the **Stream Alignment Tool** ⬚ must be engaged.

5.1.3.1   Rename a Stream Element

To rename a stream element:

1.   With the **Stream Alignment Tool** engaged right-click on the stream element you want to rename. From the shortcut menu, click **Rename Stream Element** (Figure 5.4).

2.   The **Rename Stream** dialog box will open (Figure 5.5). Enter the new name in the **Name** box (required) and a description in the **Description** box (optional). Click **OK.**

Figure 5.5 Rename Stream dialog box.

3.   Another dialog box will appear, asking the user to confirm rename. To rename, select **Yes**.

5.1.3.2   Delete a Stream Element

To delete a stream element:

1.   With the **Stream Alignment Tool** engaged right-click on the stream element you want to delete and click **Delete Stream Element** from the available shortcut menu (Figure 5.4).

2.   A dialog box will appear asking the user to confirm deletion. Click **Yes**. The stream element will be deleted from the **Map Window**.

5.1.3.3   Reverse Direction of a Stream Element

If a stream element is drawn in the wrong direction (downstream to upstream), users can reverse the direction of the stream element instead of having to delete and redraw the stream element.

To reverse direction of a stream element:

1.   With the **Stream Alignment Tool** engaged right-click on the stream element you want to reverse and click **Reverse Direction** from the available shortcut menu (Figure 5.4).

2.   A warning message will open asking if you really want to reverse the direction of the stream element. Click **Yes**. The warning message will close and the stream element's direction will be reversed.

5.1.3.4   Disconnect a Stream Element

If you want to disconnect a stream element from another stream element, FIA will allow you to do so at either the upstream or downstream end of the stream element.

To disconnect a stream element from another stream element:

1.   With the **Stream Alignment Tool** engaged right-click on the stream element you want to disconnect and either click **Disconnect Upstream** or **Disconnect Downstream** from the available shortcut menu (Figure 5.4).
2.   The stream element will now be disconnected from the stream element at the location you chose. In the **Map Window**, the upstream or downstream point will physically move away from the connecting stream element, and a stream node will remain on the connecting stream element.

5.1.3.5   Edit a Stream Element

The **Edit Stream Element** command allows you to edit the stream element description and view the stream stationing and coordinates of the upstream and downstream stream nodes, plus any stream nodes between the two.

To edit the description of a stream element:

1.   With the **Stream Alignment Tool** engaged right-click on the stream element you want to edit. From the shortcut menu, click **Edit Stream Element**. The **Stream Editor** will open (Figure 5.6).
2.   You can enter a new description or edit the existing description of the stream element in the **Description** box.
3.   The table provides you with the stream stationing and coordinates of the stream nodes that are on the selected stream element.

5.1.4   Adjusting a Stream Element

You can create, move, or delete individual vertices on a stream element. These tasks are accomplished by double clicking on a stream element while you are in the **Map Window** with the **Stream Alignment Tool** engaged.
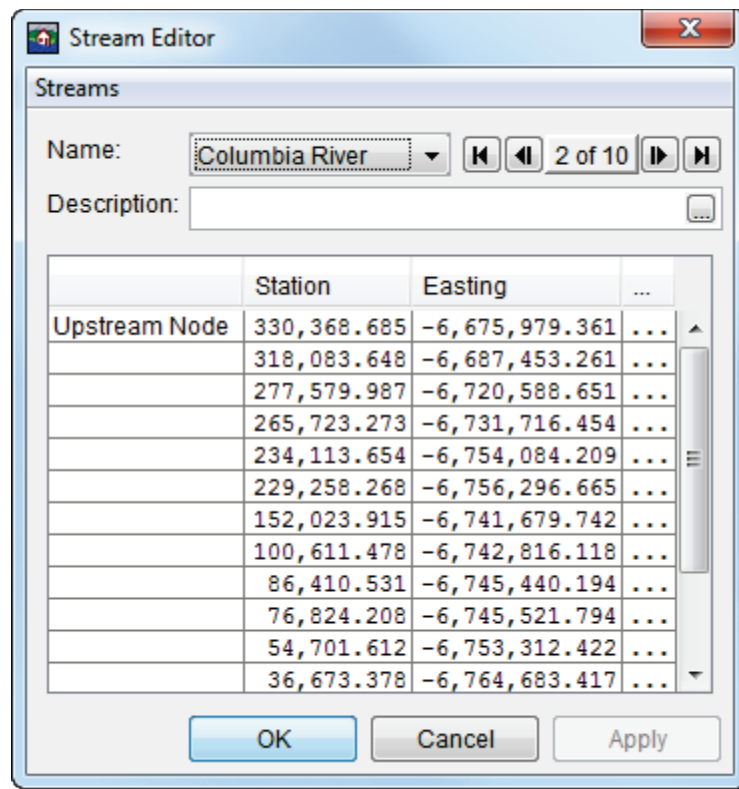
Figure 5.6 Stream Editor.

5.1.4.1   Add Vertices to a Stream Element

To add vertex points to an existing stream element:

1. With the **Stream Alignment Tool** engaged double-click on the stream element that you want to add a vertex point to. The stream element will turn red.
2. Place the pointer on the selected stream element where you want to add a vertex point. Hold down the **CTRL** key and click. A blue point will now appear on the selected stream element, which represents a new vertex point.

5.1.4.2   Move the Vertices of a Stream Element

To move vertex points on an existing stream element:

1. With the **Stream Alignment Tool** engaged double-click on the stream element on which you want to move a vertex point. The selected stream element will turn red, and the individual vertex points will turn blue.
2. Click on the vertex point you wish to move, and drag the vertex point to a new location. This is illustrated in Figure 5.7.

5.1.4.3   Delete Vertices from a Stream Element

To delete vertex points from an existing stream element:

1. With the **Stream Alignment Tool** [icon] engaged double-click on the stream element that you want to delete a vertex point. The selected stream element will turn red, and the individual vertex points will turn blue.
2. Hold down the **SHIFT+CTRL** keys, and click on the vertex point that you want to delete. The vertex point will disappear from the stream element.
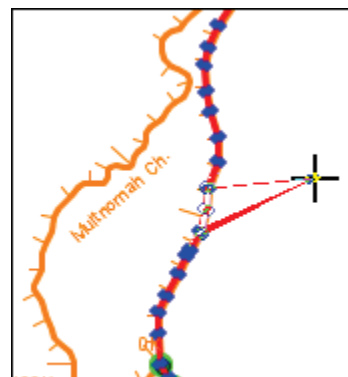
Figure 5.7 Move vertices of a stream alignment.

5.1.5   Stream Nodes
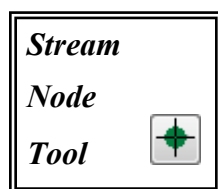
Stream Node Tool

FIA creates an upstream and downstream node for each stream element. Stream nodes are also created where stream elements meet (stream junctions). Stream nodes are used to establish stationing for a stream. To modify the default stream stationing, you must edit the stream nodes. To manipulate stream nodes you must be in the **Map Window** with the **Stream Node** tool engaged.

5.1.5.1   Add a Stream Node

As noted above, two stream nodes are created automatically – upstream and downstream nodes of a stream element. To create additional stream nodes on a stream element, using the **Stream Node Tool** [icon], place the pointer on the stream element where you want to add a stream node. Hold down the **CTRL** key and click. A circle will now appear on the stream element that indicates the new stream node.

5.1.5.2   Move a Stream Node

To move a stream node, double-click on the stream node that you want to move. It will become a green circle with a black outline and four small black dots (Figure 5.8, foreground). To move the stream node, click and

Figure 5.8 Move a Stream Node.

drag the stream node along the stream element to its new position (Figure 5.8, background).

5.1.6 Edit an Existing Stream Node

After adding stream nodes in the **Map Window**, you can delete and adjust the stationing of an individual stream node. These two tasks are accomplished through the stream node shortcut menu (Figure 5.9). To access the stream node shortcut menu, you must be in the **Map Window** with the **Stream Node Tool** ✛ engaged.
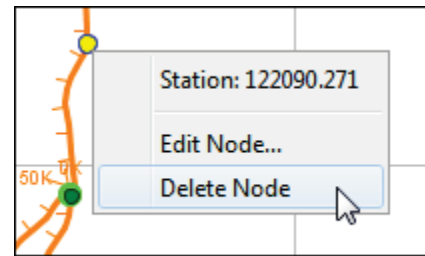
Figure 5.9 Stream Node shortcut menu.

5.1.6.1 Delete a Stream Node

To delete a stream node:

1. With the **Stream Node Tool** ✛ engaged right-click on the stream node that you want to delete and click **Delete Node** from the available shortcut menu (Figure 5.9).
2. A warning message will open asking if you really want to delete the selected stream node. Click **Yes**. The warning message will close, and the stream node will be deleted from the stream element.

5.1.6.2 Establish Stream Stationing

To establish stream stationing for a stream element, you must edit the stream nodes. To edit a stream node:

1. Right-click on the stream node that you want to edit, click **Edit Node** from the available shortcut menu, the **Stream Node Editor** will open (Figure 5.10).
2. The default stream station has the downstream node at station zero (0) and the upstream node is based on the coordinate system you used when drawing the
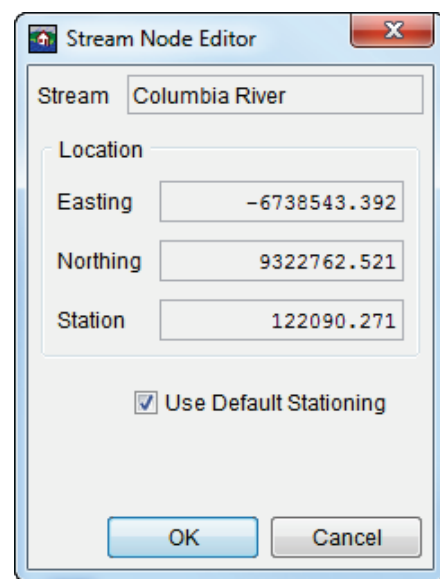
Figure 5.10 Stream Node Editor.

stream alignment. To change the stream stationing, click to clear the **Use Default Stationing** check box. The **Station** box can now be edited. Based on what you enter, FIA will automatically adjust the stream stationing for the stream element associated with the stream node. This includes the upstream and downstream nodes.

5.1.7   Stream Junctions

When two stream elements intersect, a stream junction is created. A stream junction is multiple stream nodes at one location and is represented by a dark green circle with a light green halo around the circle. You can move a stream junction and establish stream stationing at a stream junction. To manipulate stream junctions you must be in the **Map Window** with the **Stream Node Tool** ⊕ engaged.

5.1.7.1   Move a Stream Junction

To move a stream junction, double-click on the stream junction that you want to move. It becomes a green circle with a black outline and four small black dots (Figure 5.11, foreground). Click and drag the stream junction along the stream element to its new position (Figure 5.11, background).
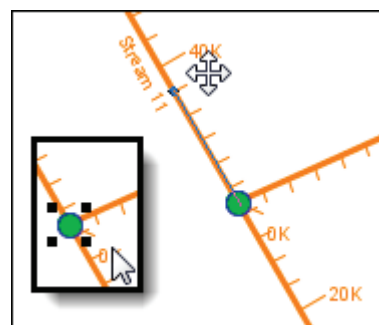
Figure 5.11 Move a Stream Junction.

5.1.7.2   Edit a Stream Junction

To edit a stream junction:

1. With the **Stream Node Tool** ⊕ engaged right-click on the stream node that you want to edit. A shortcut menu will appear (Figure 5.12).
2. Click **Edit Stream Junction**, the **Junction Editor** will open (Figure 5.13).
3. The editor displays information about the stream nodes associated with the stream

Figure 5.12 Stream Junction shortcut menu.

junction. To change the stream stationing for a stream node, clear the check box for a stream node from the **Use Default Stationing** column of the table. The cell in the **Station** column becomes

active. Based on what
you enter, FIA will
automatically adjust
the stream stationing
for the stream
associated with the
stream node.

4. Click **OK**. The
   Junction Editor will
   close.
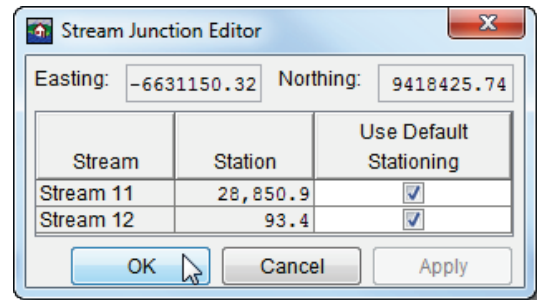
Figure 5.13 Stream Junction Editor.

5.1.8   Change Stream Alignment Properties

Stream alignment properties that you can change include color, width,
font, and tic mark properties. Also, you can change stream node
properties, stream junction properties, and set the visualization scale for
the stream alignment. Stream alignment properties are changed from the
Stream Alignment Properties dialog box (Figure 5.14).
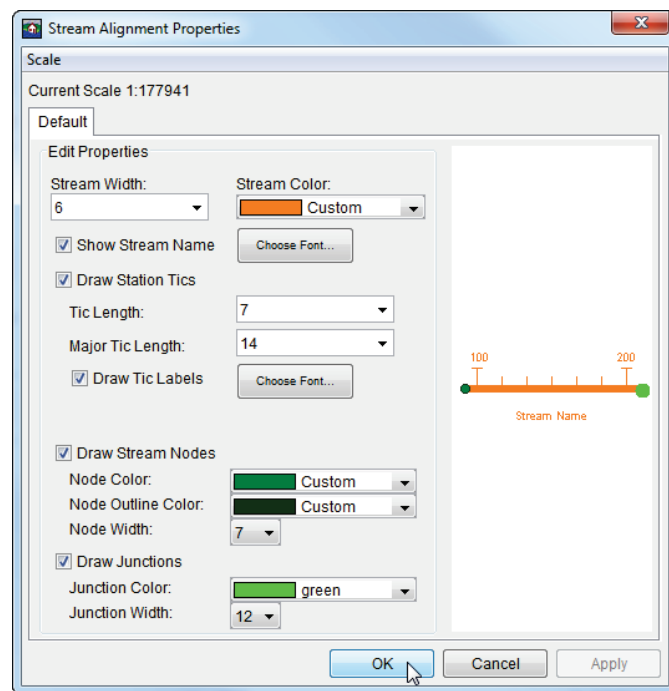
Figure 5.14 Stream Alignment Properties dialog box.

To access the **Stream Alignment Properties** dialog box:

1. Starting in the **Content Pane**, select the **Maps** tab. Under the **Map
   Layers** folder, locate **Stream Alignments** and click (the name will
   highlight in blue). Right-click and from the shortcut menu (Figure
   5.15) select **Properties**.

2. The **Stream Alignment Properties** dialog box will open.

5.1.8.1 Stream Properties

Stream properties that you can change include color, width, font, and tic mark properties. These changes are made to all stream elements for the stream alignment from the **Stream Alignment Properties** dialog box (Figure 5.14):
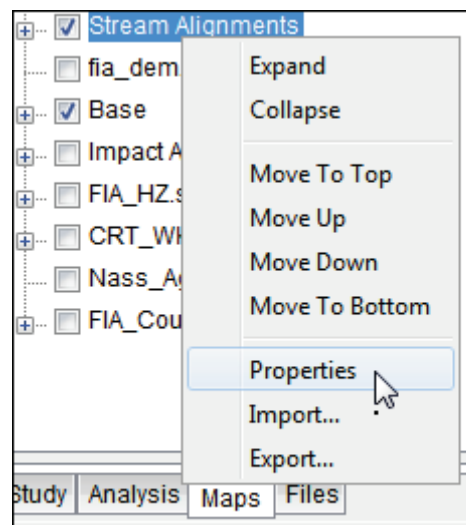
Figure 5.15 Stream Alignments shortcut menu (Content Pane).

1. To change the width of all the stream elements in the stream alignment, select a width from the **Stream Width** list (Figure 5.16) or enter a number.
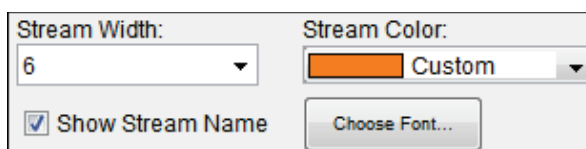
Figure 5.16 Stream Width list (Stream Alignment Properties dialog box).

2. Changes to the stream elements can be previewed in the example to the right of the **Edit Properties** box (Figure 5.17). For the changes to appear in the **Map Window**, click **Apply**, and the **Map Window** will update.
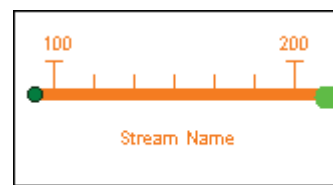
Figure 5.17 Formatting preview (Stream Alignment Properties dialog box).

3. To change the color of all the stream elements in the stream alignment, select a pre-defined color from the **Stream Color** list (Figure 5.14). The default color is orange, and there are several colors to choose from. Users can also create their own colors from a color chooser. From the **Stream Color** list, click **Custom**, the **Choose a Color** dialog box (Figure 5.18) will open. See Appendix F for further details on the color chooser.

4. To change the font for the stream names in the stream alignment, click **Choose Font** (Figure 5.14). A **Font Chooser** will open (Figure 5.20). From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, from the **Type** list select a font from the available list. Use the **Size** slider to change the size of

the font. As you change the font attributes, the current selection is shown in the box at the top of the **Font Chooser**.

5. If you do not want the stream names to display on the stream alignment, clear **Show Stream Name** (Figure 5.14).
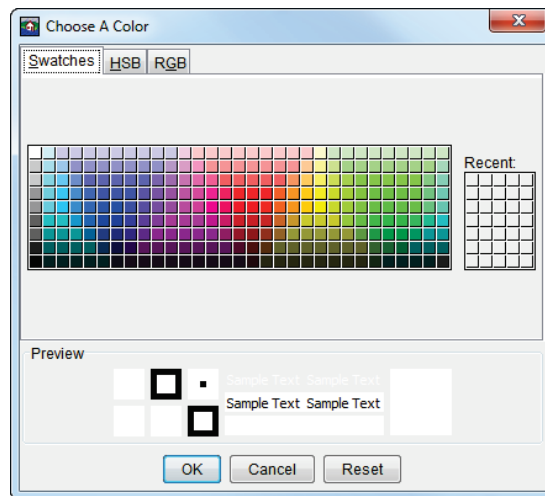
Figure 5.18 Choose a color dialog box.

Tic marks appear on a stream representing the stream stationing of the stream. There are two types of tic marks – minor and major, to change the appearance of tic marks in the stream alignment:

1. If you do not want the station tic marks to display on the stream alignment clear **Draw Station Tics** (Figure 5.19).
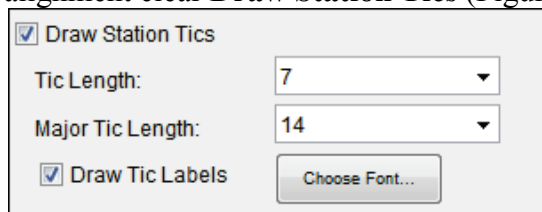
Figure 5.19 Tic drawing properties (Stream Alignment Properties dialog box).

2. To change the minor tic lengths, from the **Tic Length** list select a length or enter a number. To change the major tic lengths, from the **Major Tic Length** list select a length or enter a number.

3. To change the font for the tic mark labels in the stream alignment, click **Choose Font** (Figure 5.19). A **Font Chooser** will open (Figure 5.20). From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, from the

Figure 5.20 The Font Chooser.

Type list select a font from the available list. Use the **Size** slider to change the size of the font. As you change the font attributes, the current selection is shown in the box at the top of the **Font Chooser**.

4. If you do not want the tic mark labels to display on the stream alignment, clear **Draw Tic Labels** (Figure 5.19).

5. To display any of the changes made in steps 1 - 4 to the stream alignment, click **Apply** (which will change the stream alignment on the **Map Window** but will not close the **Stream Alignment Properties** dialog box) or **OK** (which will change the stream alignment on the **Map Window** and will close the **Stream Alignment Properties** dialog box).

5.1.8.2   Stream Node Properties

Stream node properties that you can change include color, width, and whether or not to display the stream nodes on the stream alignment. From the **Stream Alignment Properties** dialog box (see Figure 5.14, page 5-11):

1. To change the width of all the stream nodes in the stream alignment, select a width from the **Node Width** list (Figure 5.21).

2. Changes to the stream nodes can be previewed in the example to the right of the **Edit Properties** box (see Figure 5.17, page 5-12). For the changes to appear in the **Map Window**, click **Apply**, and the **Map Window** will update.
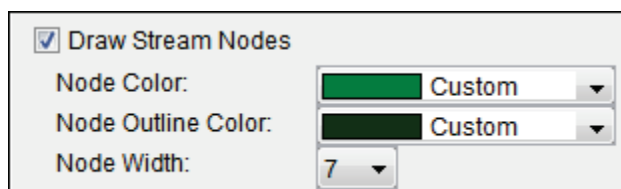
Figure 5.21 Stream Node drawing properties (Stream Alignment Properties dialog box).

3. To change the fill color of all the stream nodes in the stream alignment, select a pre-defined color from the **Node Color** list (Figure 5.21). To change the outline color of all the stream nodes in the stream alignment, select a pre-defined color from the **Node Outline Color** list (Figure 5.21). The default color is green, there are several colors to choose from, plus you can create your own colors from a color chooser. From the **Node Color** list or the **Node Outline Color** list, click **Custom**, the **Choose A Color** dialog box (Figure 5.18) will open. See Appendix F for further details on the color chooser.

4. If you do not want the stream nodes to display on the stream alignment, clear **Draw Stream Nodes** (Figure 5.21).

5. To display any of the changes made in steps 1 - 4 to the stream alignment, click **Apply** (which will change the stream alignment on the **Map Window** but will not close the **Stream Alignment Properties** dialog box) or **OK** (which will change the stream alignment on the **Map Window** and will close the **Stream Alignment Properties** dialog box), from the **Stream Alignment Properties** dialog box (see Figure 5.14, page 5-11).

5.1.8.3   Stream Junction Properties

Stream junction properties that you can change include color, width, and whether or not to display the stream junctions on the stream alignment. From the **Stream Alignment Properties** dialog box (see Figure 5.14, page 5-11):

1. To change the width of all the stream junctions in the stream alignment, select a width from the **Junction Width** list (Figure 5.22)
2. Changes to the stream junctions can be previewed in the example to the right of the **Edit Properties** box (see Figure 5.17, page 5-12). For the changes to appear in the **Map Window**, click **Apply**, and the **Map Window** will update.
3. To change the color of all the stream junctions in the stream alignment, select a pre-defined color from the **Junction Color** list (Figure 5.22). The default color is green, there are several colors to choose from, plus you can create your own colors from a color chooser. From the **Junction Color** list, click **Custom**, the **Choose A Color** dialog box (see Figure 5.18, page 5-13) will open. See Appendix F for further details on the color chooser.

Figure 5.22 Stream Junctions drawing properties.

4. If you do not want the stream junctions to display on the stream alignment, clear **Draw Junctions** (Figure 5.22).
5. To display any of the changes made in steps 1 - 4 to the stream alignment, click **Apply** (which will change the stream alignment on the **Map Window** but will not close the **Stream Alignment Properties** dialog box) or **OK** (which will change the stream alignment on the **Map Window** and will close the **Stream Alignment Properties** dialog box), from the **Stream Alignment Properties** dialog box (see Figure 5.14, page 5-11).

5.1.8.4   Visualization Scale

You can set the visualization scale for the stream alignment from the **Stream Alignment Properties** dialog box (see Figure 5.14, page 5-11). Setting the visualization scale will allow you to make the stream alignment visible and not visible as you zoom in and out of the **Map Window**.

1. To set the visualization scale for the stream alignment to a default, from the **Scale** menu, click **Set to Defaults**. The visualization scale is set automatically.
2. To enter your own scale factor, from the **Scale** menu, click **New**. The **Create new Scale** dialog box opens (Figure 5.23), in the **Scale** box, enter your values for the scale factor.
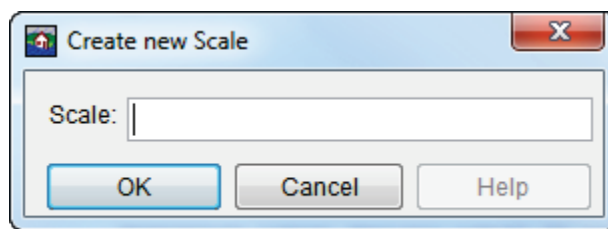
Figure 5.23 Create New Scale dialog box.

3. Click **OK**. The **Create new Scale** dialog box will close, and the visualization scale will be set for the **Stream Alignment Layer**.
4. A new tab will appear in the **Stream Alignment Properties** dialog box for each scale you create. Users can set separate drawing properties for each scale.

5.1.9 Import a Stream Alignment from Shapefile

To import a stream alignment, you must have an active map layer that is a shapefile (see Chapter 3, Section 3.1.1, page 3-2) that represents the stream system of your watershed (**Figure 5.24**). This shapefile must have an attribute that is the name of each stream, and each stream must form a contiguous line with one upstream point and one downstream point.

To import a stream alignment:

1. Starting in the **Study Pane**, under the **Watershed** folder of the **Study Tree** right-click **Stream Alignments**.
2. From the shortcut menu, select **Import**.
3. The **Import Stream Alignment** dialog box will open (Figure 5.25).
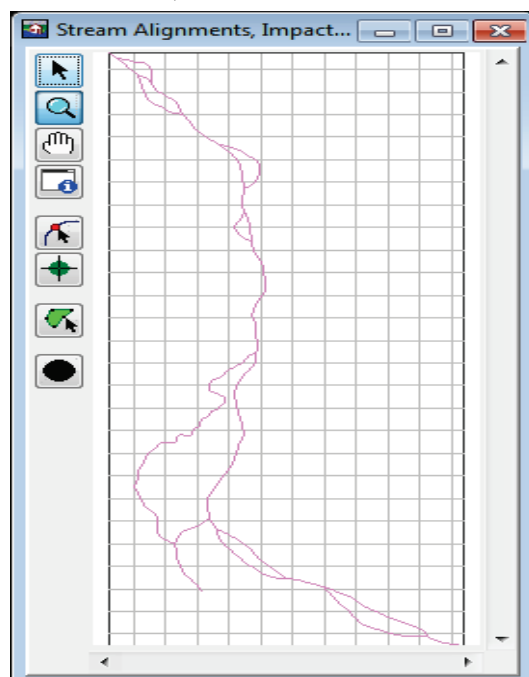
Figure 5.24   Shapefile that represents the stream system of a watershed.
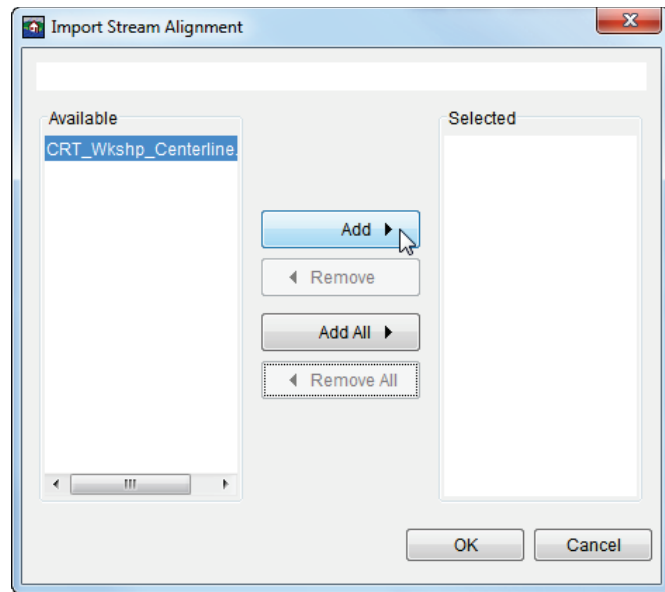
Figure 5.25 The Import Stream Alignment dialog box.

4.   The **Available** box lists the available polyline shapefiles in your study that represent stream systems. Click on the file you would like to use to import the stream alignment, and it will be highlighted in blue. Click **Add**. The shapefile name will now appear under the **Selected** box.

5.   To import from multiple shapefiles, click on each file while holding down control and then click **Add.** If you would like to add all the files, click **Add All**. Should you wish to remove a selected shapefile, select it under the **Selected** box and click **Remove**.

6.   Once the correct shapefile(s) have been selected, click **OK**. The second **Import Stream Alignment** dialog box will open (Figure 5.26).
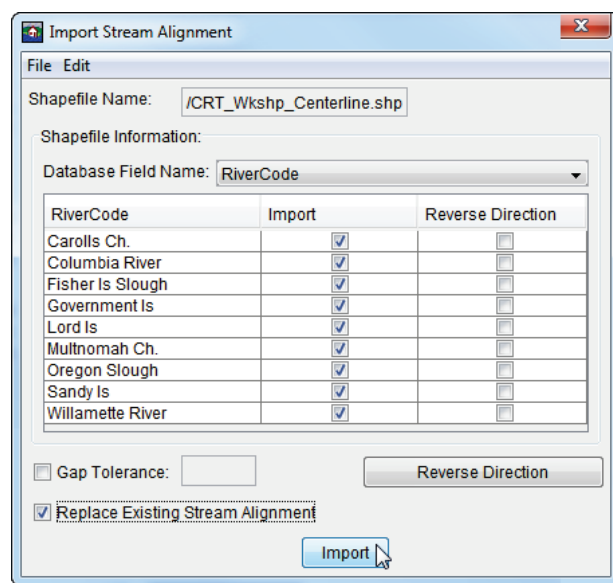
Figure 5.26 Another Import Stream Alignment dialog box.

7. From the **Database Field Name** dialog box, select the shapefile field that represents the river names for the stream system (e.g. *RiverCode* in Figure 5.26). When you have selected the correct database field name, the table will list the stream names. If each stream is a contiguous line with one upstream point and one downstream point, then the checkbox in the **Import** column of the table will be set. If there is something wrong with the line that represents a stream, the checkbox will not be set, and FIA will not import that stream element.

8. **Gap Tolerance** is used to connect stream elements that have a gap between the end point and the junction with another stream. By default, the **Gap Tolerance** is set to zero (0.0) and grayed out, so FIA can connect streams to other streams that are right on top of each other. If you click **Gap Tolerance**, then you need to enter a value larger than zero in the **Gap Tolerance** box (Figure 5.26), which will allow FIA to connect streams that have a small gap between them.

9. The **Replace Existing Stream Alignment** flag allows you to replace the existing stream alignment with the one you are currently importing from the selected shapefile.

10. Once everything is set, click **Import**. The stream alignment will be automatically drawn in the **Map Window** and the **Import Stream Alignment** dialog box will close (Figure 5.27). A message box will appear notifying you that your stream alignment has been successfully imported.
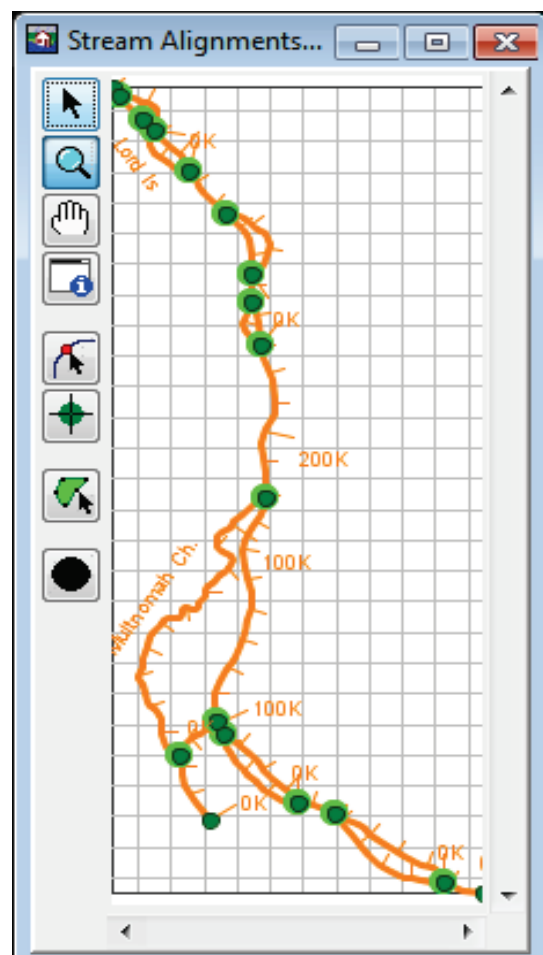
Figure 5.27 Successfully imported stream alignment (cf. Figure 5.24, page 5-16).

5.1.10   Import Existing Stream Alignment

If users have already built and saved a stream alignment in FIA or other HEC software, they may add them to a study without using a shapefile.

To add an existing stream alignment:

1.   Starting in the **Study Pane**, under the **Watershed** folder of the **Study Tree** right-click **Stream Alignments**.
2.   From the shortcut menu, select **Import**.
3.   The **Confirm Import** dialog box will appear, warning the user that import will replace all stream elements, junctions and nodes with the new file and asking for confirmation. To continue, select **Yes**.
4.   An **Open Browser** will open. Browse to the location of the stream alignment (e.g. *stream.align*) you wish to import (Figure 5.28).
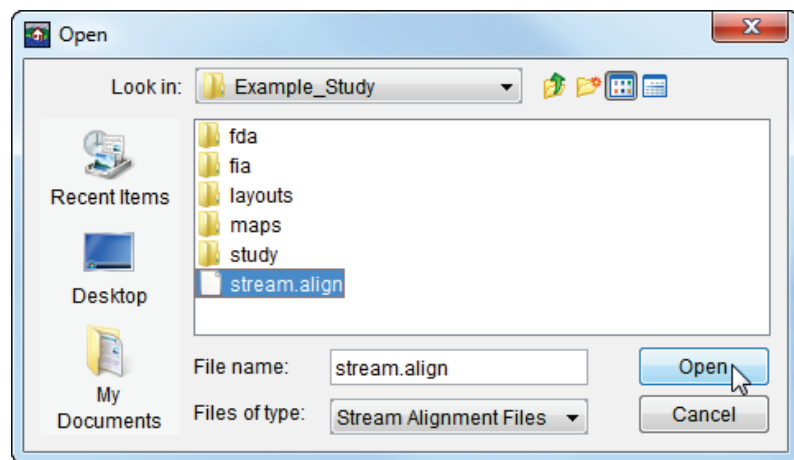
Figure 5.28 Use the Open Browser to locate the stream alignment you would like to import.

5.   Once the correct file is selected, click **Open**.
6.   The stream alignment will now display in the **Map Window**.

5.1.11   Export a Stream Alignment

If you have a digitized stream alignment, you can save that stream alignment as a shapefile. To export a stream alignment you must have a stream alignment in the **Map Window**.

To export a stream alignment:

1.   Starting in the **Content Pane**, under the **Maps** tab click on **Stream Alignments.** It will be highlighted in blue. Right-click and from the shortcut menu select **Export**.
2.   The **Save** browser will open (Figure 5.29). Browse to the location you would like to save the file. Enter a name in the **File name** box.
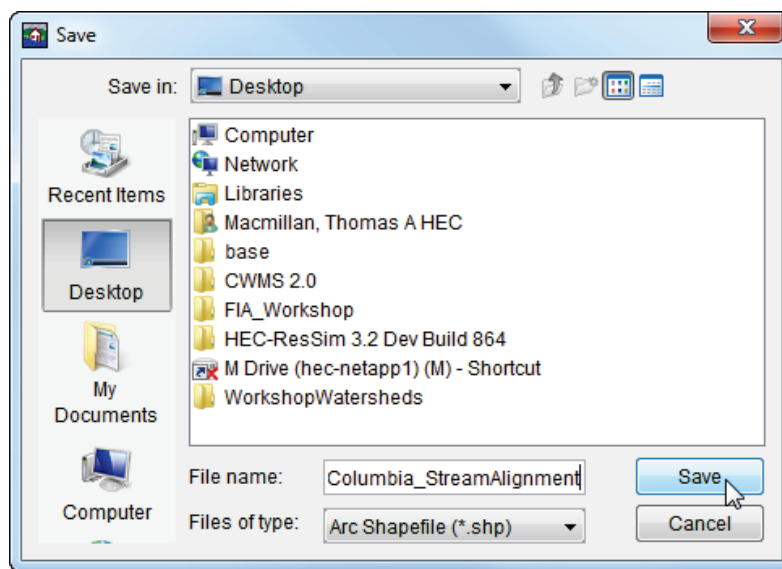
Figure 5.29 Save browser.

Click **Save**. The **Save** browser will close and an ESRI shapefile of the displayed stream alignment will be saved in the location you have selected.

5.1.12   List of Streams

FIA allows you to view the list of streams that make up your stream alignment. To view the **List of Streams**:

1. Starting in the **Study Pane**, under the **Watershed** folder, right click on **Stream Alignments**. From the shortcut menu, select **List of Streams**.
2. The **List of Streams** dialog box will open (Figure 5.30).
3. The report provides a list of streams that have been input for the stream alignment, along with the stream's description.
4. From the **File** menu, you can send the report to a printer, or you can export the report to an ASCII tab delimited file.
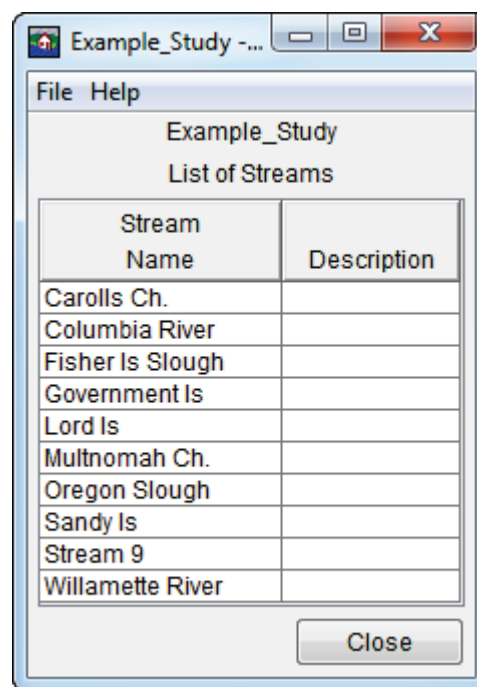
Figure 5.30 List of Streams.

5. When finished viewing the report, click **Close**. The **List of Streams** dialog box will close.

5.2   Create Impact Areas

The next step in defining the common schematic is to create your impact areas and projects. In HEC-FIA, an **impact area** is the basic unit of analysis and indicates areas that have the potential to be affected by rising river or reservoir stages.

An **impact area** is any distinct portion of a watershed that is affected by rising stage in a stream, river, lake, or reservoir. The boundaries of an impact area are defined by a closed polygon, and you characterize the effects of rising stage by describing the property and population at risk in the area bounded by the polygon.

You can use underlying map layers to aid in placing impact areas at the proper location in the **Map Window**. Besides location, a map layer can also give you a concept of the dimensions of an impact area.

5.2.1   Create an Impact Area

Impact Area Polygon Tool	
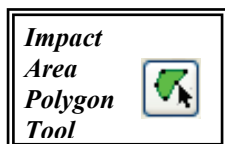
You can either digitize your impact areas by hand or import impact areas from a shapefile. The following are the steps for digitizing impact areas; see Section 5.2.4 (page 5-26) for details on importing impact areas.

To add an impact area to the **Map Window**:

1. You must be in the **Map Window**.
2. On the map toolbar, press the **Impact Area Polygon Tool**.
3. Hold down the **CTRL** key while clicking the mouse button. Each click creates a vertex point (Figure 5.31), which is the boundary
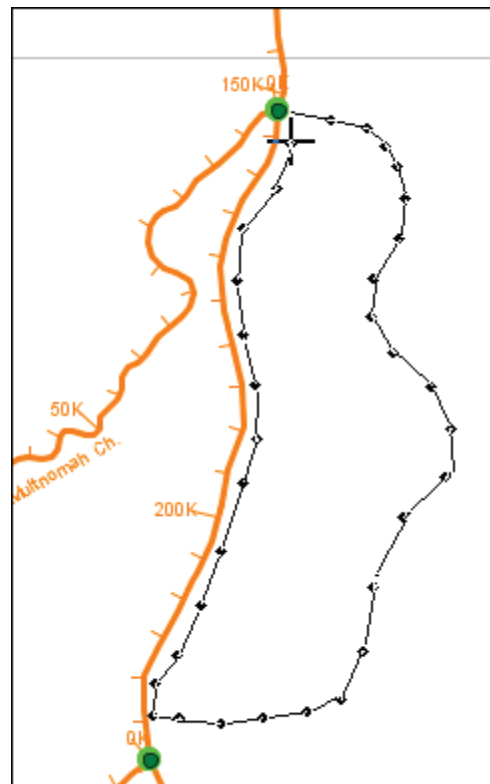
Figure 5.31 Digitizing an Impact Area.

of the polygon. When reaching the end point, release the **CTRL** key and click.

Figure 5.32 Name New Impact Area dialog box.

4. The **Name New Impact Area** dialog box will open (Figure 5.32). You must enter a name, and if needed, you can enter a description. Click **OK**. The **Name New Impact Area** dialog box will close.

5. The impact area will now appear in the **Map Window** as a green polygon with a name. Repeat steps 3 and 4 for each impact area in your study.

5.2.2   Edit an Existing Impact Area

After drawing an impact area in the **Map Window**, you can rename an individual impact area, delete individual impact areas, create configuration notes for individual impact areas, and edit individual impact areas properties. All of these tasks are accomplished through the impact area shortcut menu (Figure 5.33). To access the impact area shortcut menu, you must be in the **Map Window** with the **Impact Area Polygon Tool** ![icon] engaged.
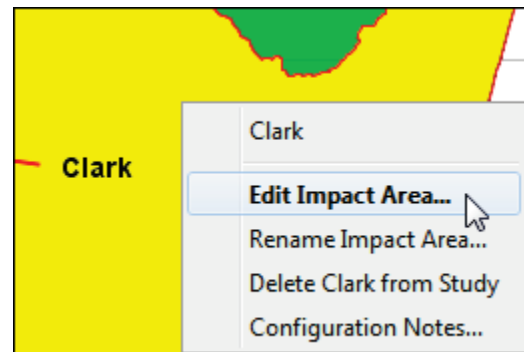
Figure 5.33 Impact Area shortcut menu.

5.2.2.1   Rename an Impact Area

To rename an impact area:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Impact Area Polygon Tool** ![icon].
3. Right-click on the impact area you want to rename, and click **Rename Impact Area** from the available shortcut menu (Figure 5.33).

4. The **Rename Impact Area** dialog box will open (Figure 5.34). Enter the new **Name** (required) and **Description** (optional) for the impact
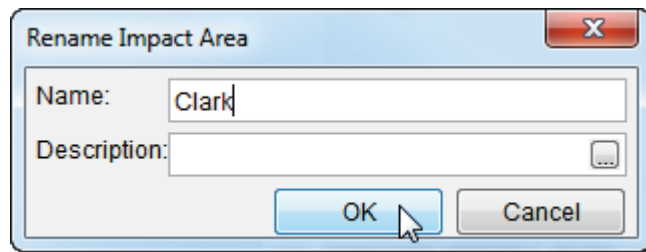
Figure 5.34 Rename Impact Area dialog box.

area. Since this name creates the "B-Part" of a DSS pathname, FIA will substitute an underline for the following characters: **? \ / * " < > | & ; ' [ ] ( ) : % ! ~**. Also, the impact area name must be less than sixty-four (64) characters.

5. Click **OK**. A warning message will open asking you if you really want to rename the selected impact area. Click **Yes**. The warning message and the **Rename Impact Area** dialog boxes will close, and the impact area will display the new name.

5.2.2.2   Delete an Impact Area

To delete an impact area:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Impact Area Polygon Tool** .
3. Right-click on the impact area you want to delete, and click **Delete Impact Area** from the available shortcut menu (Figure 5.33).
4. A warning message will open asking you if you really want to delete the selected impact area. Click **Yes**. The warning message will close.

5.2.2.3   Add Configuration Notes for an Impact Area

Configuration notes allow you to add information about an impact area specific to a configuration. These are useful for describing impact areas that belong to configurations that include non-structural measures. If there are non-structural attributes for an impact area, you can have a symbol appear in the **Map Window** for the selected impact area. Configurations notes are only added when other configurations have been defined. You do not need configuration notes for the default configuration Study. For further information on configurations, see Section 5.3.

To add configuration notes for an impact area:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Impact Area Polygon Tool** .

3. Right-click on the impact area to which you want to add configuration notes, and click **Configuration Notes** from the available shortcut menu (Figure 5.33).
4. The **Configuration Notes Editor** will open (Figure 5.35). The **Configuration** box displays the current configuration. Enter information about this impact area for the current configuration in the **Configuration Notes** box.
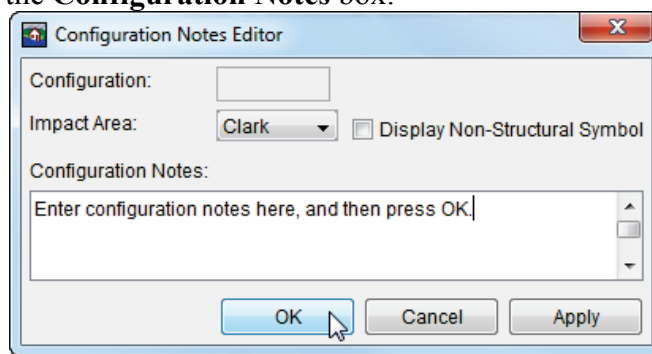
Figure 5.35 Configuration Notes Editor.

5. If the notes are in regards to non-structural attributes, select **Display Non-Structural Symbol**. This will cause a symbol to appear in the selected impact area on the **Map Window**.

5.2.2.4   Edit Impact Area Properties

In the **Map Window**, you describe the general and aesthetic properties of an impact area. When you create alternatives, you will enter detailed information for each impact area in the **Impact Area Editor** (Chapter 9, Section 9.7, page 9-8). The impact area properties that you can edit in the **Map Window** include: the description of the impact area, the stream associated with the impact area, what bank of the stream the impact area is on, the color of the impact area polygon, the position of the impact area label on the **Map Window**, the geographical location of the index station, and the geographical location of the beginning and ending stations of the impact area polygon.

To edit impact area properties:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Impact Area Polygon Tool** .
3. Right-click on the impact area whose properties you would like to change, and click **Edit Impact Area** from the available shortcut menu (see Figure 5.33).
4. The **Impact Area Properties Editor** will open (Figure 5.36).
5. If the **Auto** check box is selected for an item, the value for this data item is automatically updated as the polygon or index location is moved in the **Map Window**. By default, when a polygon is drawn in the **Map Window**, these values are filled in automatically. If the
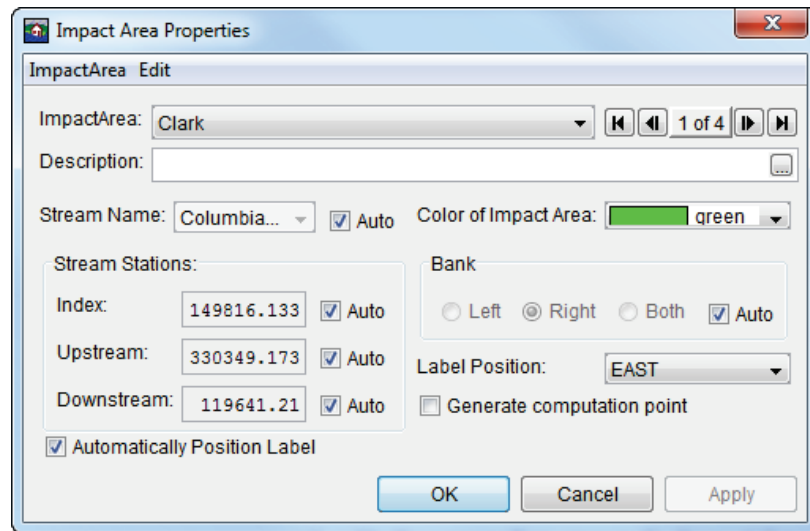
Figure 5.36 Impact Area Properties dialog box.

check box is cleared, data can be entered manually into these fields, and the value will cease to be automatically updated if changes are made to the polygon. However, reselecting the check box allows the values to be automatically determined when edits are made in the **Map Window**.

6. From the **Description** box (Figure 5.36) you can edit or add a description.

7. If the default stream name selected by FIA is incorrect, clear the **Auto** check box, and from the **Stream Name** list select the appropriate stream name.

8. If the default bank selected by FIA is incorrect, clear the **Auto** check box, and select the appropriate stream bank from the **Bank** box.

9. By default, the color of the impact area polygon that you have drawn is green. You can change the color by selecting a different color from the **Color** list.

10. The **Label Position** list allows you to place the impact area label on the **Map Window**.

11. For each impact area, FIA generates an index station; this index station is a location. If you want to place the index station yourself, clear the **Auto** check box next to the **Index Station** box, and then enter the new stream stationing in the **Index Station** box.

12. Each impact area has a beginning and ending station that the program automatically computes. If you want to change the stream stationing the program has generated, clear the **Auto** check box next to the **Upstream** and **Downstream** boxes, and enter the new stream stationing in the **Upstream** box and the **Downstream** box.

5.2.3   Edit Impact Area Drawing Properties

You can override the default font used in drawing an impact area. To edit impact area drawing properties:

1.   Starting in the **Content Pane**, select the **Maps** tab. Under the **Map Layers** folder, locate **Impact Area Polygons** and click (the name will highlight in blue). Right-click and from the shortcut menu select **Properties**.
2.   The **Impact Area Properties** dialog box will open.
3.   Click the **Impact Area** tab; to turn off impact area names on the **Map Window**, clear the **Draw Impact Area Name** check box.
4.   To change the font, size, and style of an impact area name, click **Name Font**. A **Font Chooser** will open (Figure 7.8).
5.   From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, from the **Type** list select a font from the available list. Use the **Size** slider to change the size of the font. As you change the font attributes, the current selection is shown in the box at the top of the **Font Chooser**.

5.2.4   Import Impact Areas

To import a stream alignment, you must have an active map layer that is a shapefile (see Chapter 3, Section 3.1.1, page 3-2) representing the impact areas that are in your study (e.g. the counties shapefile in Figure 5.37). This shapefile must have an attribute that is the name of each impact area, and each impact area must form a contiguous polygon.

To import impact areas:

1.   Starting in the **Study Pane**, under the **Watershed** folder of the **Study Tree** right-click **Impact Area Polygons**.
2.   From the shortcut menu, select **Import**.
3.   The **Import Impact Areas** dialog box will open (Figure 5.38).
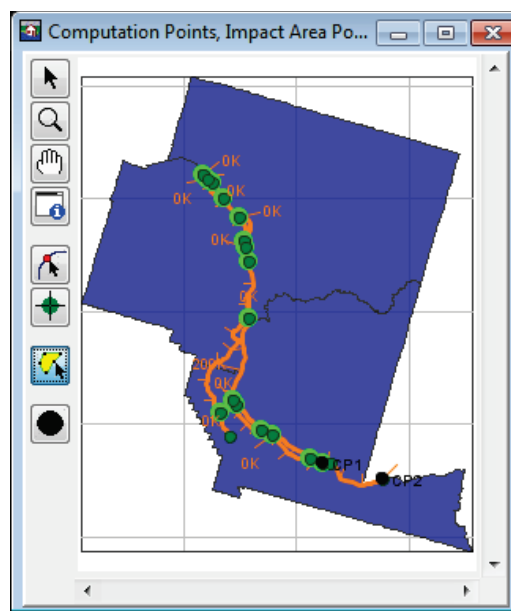
Figure 5.37 ESRI shapefile representing the Impact Areas for a given study.
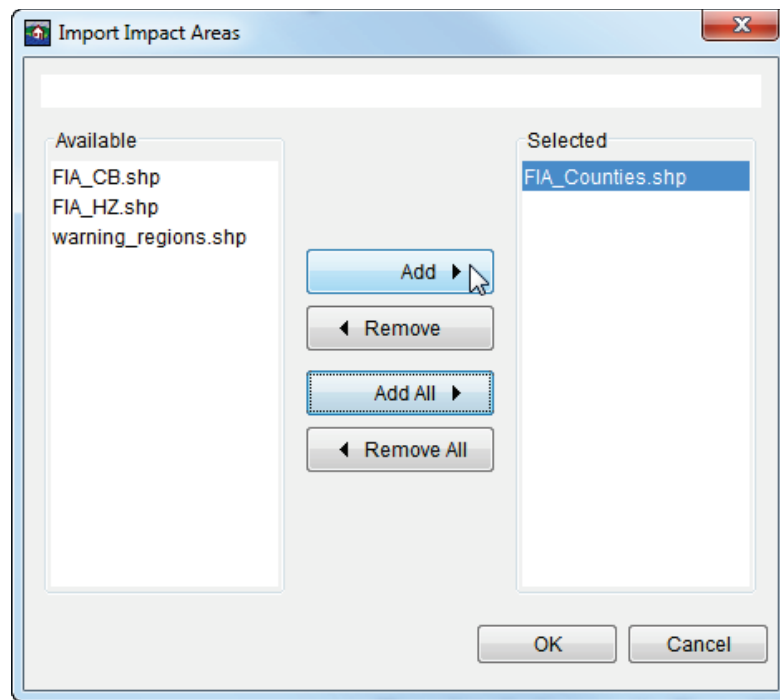
Figure 5.38 Import Impact Areas dialog box.

4. The **Available** box lists the available polyline shapefiles in your study that could represent impact areas. Click on the file that represents the impact area for your study, and it will be highlighted in blue. Click **Add**. The shapefile name will now appear under the **Selected** box. In Figure 5.38, *FIA_Counties.shp* has been selected to import impact areas from counties.

5. To import from multiple shapefiles, click on each file while holding down the control key and then click **Add.** If you would like to add all the files, click **Add All**. Should you wish to remove a selected shapefile, select it under the **Selected** box and click **Remove**.

6. Once the correct shapefile(s) have been selected, click **OK**. The second **Import Impact Areas** dialog box will open (akin to Figure 5.26). However, no impact areas will appear to import. You must select these in the **Map Fields** dialog box.

7. In the upper right-hand corner of this dialog, click **Map Fields**. The **Map Fields** dialog box will open (Figure 5.39).
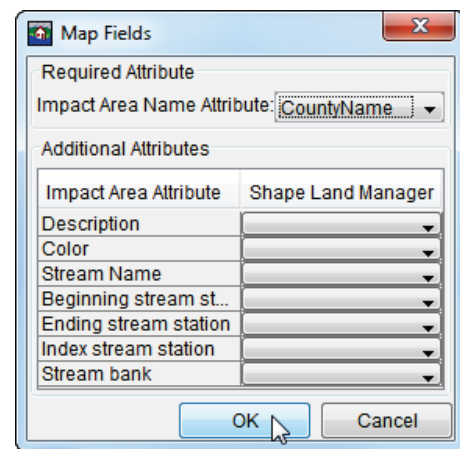
Figure 5.39 Map Fields dialog box.

8. From the **Impact Area Name Attribute** dropdown list, select the shapefile attribute field that represents the name of each impact

area. In the example in Figure 5.39, this attribute is *CountyName.* You can select optional attributes from the dropdown lists below.

9. Click **OK**. The **Map Fields** dialog box will close and you will be back in the **Import Impact Areas** dialog box. The names of each impact area will now display to the left (Figure 5.40).
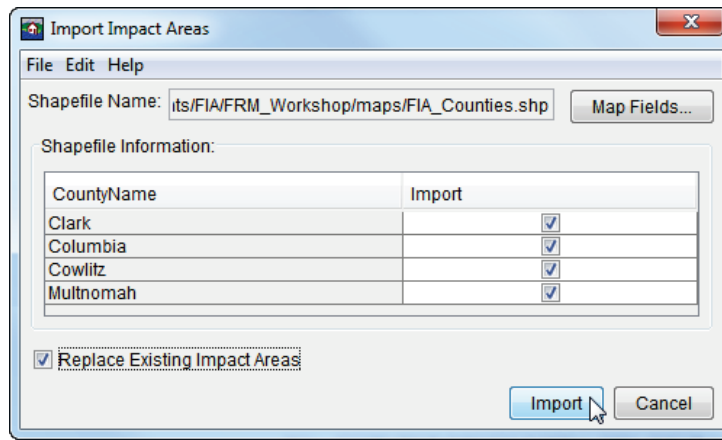
Figure 5.40 Import Impact Areas dialog box with name field selected in the Map Fields dialog box.

10. If there is something wrong with the polygon that represents an impact area, the checkbox will not be set, and FIA will not import that stream element.

11. The **Replace Existing Impact Area** flag allows users to replace the existing impact area with the one you are currently importing from the selected ESRI shapefile.

12. Once everything is set, click **Import**. The stream alignment will be automatically drawn in the **Map Window** and the

Figure 5.41 Successfully imported Impact Area Polygons.

Import Stream Alignment dialog box will close and a progress box will appear. When import is complete, a message box will appear notifying you that your stream alignment has been successfully imported.

13. All the dialogs will close and your impact area polygons will display in the **Map Window** (Figure 5.41).

5.2.5   Export Impact Areas

If you have digitized impact areas, you can save the impact areas as an ESRI shapefile. To export impact areas you must have impact areas in the **Map Window**.

To export impact areas:

1. Starting in the **Study Pane**, under the **Watersheds** folder, right-click on **Impact Area Polygons.** From the shortcut menu select **Export**.
2. The **Save** browser will open (see Figure 5.29, page 5-20). Browse to the location you would like to save the file. Enter a name in the **File name** box. Click **Save**. The **Save** browser will close and an ESRI shapefile of the displayed stream alignment will be saved in the location you have selected.

5.2.6   List of Impact Areas

FIA allows you to view a comprehensive list of the impact areas in your watershed. To view the **List of Impact Areas**:

1. Starting in the **Study Pane**, under the **Watersheds** folder, right-click on **Impact Area Polygons.** From the shortcut menu select **List of All Impact Areas**. The **List of All Impact Areas** dialog box will appear (Figure 5.42).
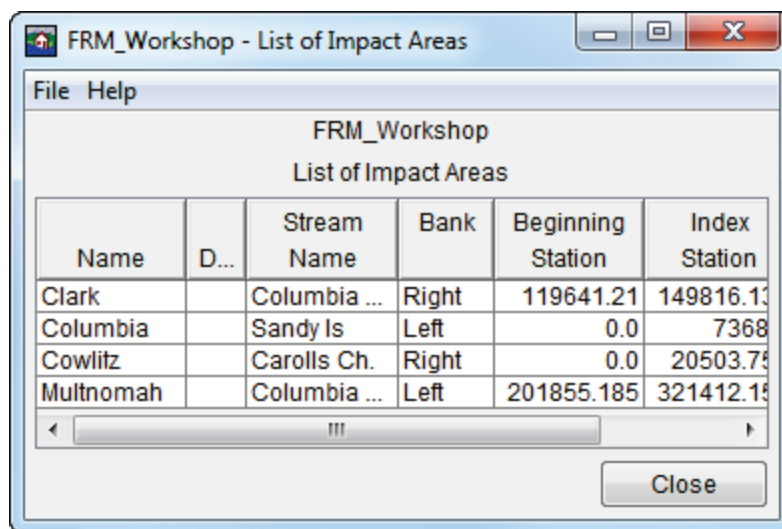
Figure 5.42 List of All Impact Areas dialog box.

2. The report provides a list of impact areas that have been input for the watershed, along with the impact area's description, and location (stream name, bank, and station data).

3. From the **File** menu, you can send the report to a printer, or you can export the report to an ASCII tab delimited file.
4. When done viewing the report, click **Close**. The **List of Impact Areas** dialog box will close.

5.3   Configurations

A **configuration** is a specific physical arrangement of projects that will be modeled.

Once you have added all of the reservoirs, levees, and computation points needed for the watershed, you will need to create one or more configurations. For HEC-FIA, a **configuration** is a specific physical arrangement of projects that will be modeled. Users add multiple configurations to model different arrangements of projects. For example, a modeling team might create an *Existing Condition* configuration that contains existing levees and a *Future Condition* that contains added levees and a reservoir. The project tools that create reservoirs, levees, and other projects will not appear until a configuration is created.

5.3.1   Create a Configuration

To create a configuration:

1. Starting in the **Study Pane**, under the **Watershed** folder, right-click on **Project Configurations**. *From the shortcut menu, select New (Figure 5.43).*
2. The **Create New Project Configurations** dialog box will appear.
3. In the **Name** box enter a name for your configuration (required) and in the **Description** box, enter a short description (optional).
4. FIA will automatically select a directory in which to create the new configuration.
5. Once you have completed the preceding steps, select **OK**.
6. If there are projects in your existing configuration, the **Select Map** dialog box will open (**Figure 5.44**). Select **Existing Map Window** to copy all of your existing projects (reservoirs, levees, channel modifications, diversions, etc. created in the default configuration) into your new configuration.
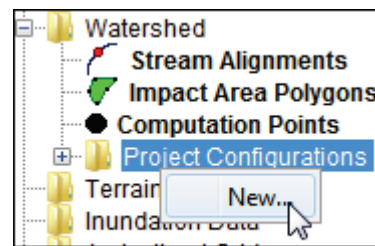
Figure 5.43 Project Configurations folder shortcut menu.
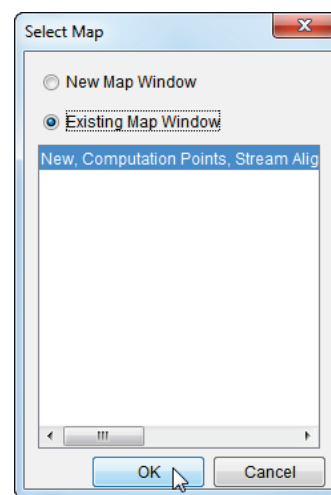
Figure 5.44 Select Map dialog box.

Select **New Map Window** to create a new configuration without projects.

5.3.2   Rename a Configuration

To rename a configuration:
1. Starting in the **Study Pane**, under the **Watershed** folder, expand the **Project Configurations** subfolder and right-click the name of the configuration you wish to rename. From the shortcut menu, select **Rename** (**Figure 5.45**).
2. The **Rename Project Configurations** dialog box will appear.
3. Enter a new name (required) and a short description (optional).
4. The **Confirm Rename** dialog box will appear, asking the user to confirm renaming of the configuration.
5. To rename, select **Yes.** The dialog box will close and the new name of your configuration will appear under the **Project Configurations** folder in the **Study Tree**.
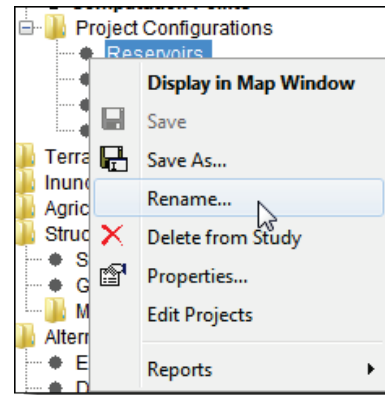
Figure 5.45 Configurations shortcut menu.

5.3.3   Delete a Configuration

From the configurations shortcut menu (**Figure 5.45**), users can also delete a configuration:

1. Starting in the **Study Pane**, under the **Watershed** folder, expand the **Project Configurations** subfolder and right-click the name of the configuration you wish to delete. From the shortcut menu, select **Delete from Study** (**Figure 5.45**).
2. The **Confirm Manager Delete** dialog box will appear, asking the user if they are sure they want to delete the configuration.
3. To delete, select **Yes**. The configuration will be removed from the study and the hard drive.

5.3.4   Save a Configuration As

The **Save As** function can be useful if users want to make a slightly-modified version of a particular configuration. A copy of the configuration will be made with a different name.

To save a configuration under a new name:

1. Starting in the **Study Pane**, under the **Watershed** folder, expand the **Project Configurations** subfolder and right-click the name of the configuration you wish to copy. From the shortcut menu, select **Save As** (**Figure 5.45**).
2. The **Save Configurations As** dialog box will appear (**Figure 5.46**).
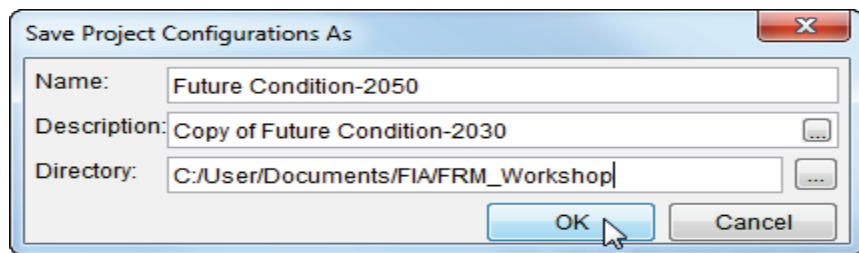
Figure 5.46 Save Configurations As dialog box.

3. Enter a new name (required) and a short description (FIA will enter an automatic description including the name of the file that is being copied).
4. Click **OK**. The **Save As** dialog box will close, and the new name of your configuration will appear in the **Study Tree**.

5.3.5   Edit All Projects

Users may edit a few key properties of each project in a configuration using the **All Projects Editor**.

To edit all projects within a configuration:

1. Starting in the **Study Pane**, under the **Watershed** folder, expand the **Project Configurations** subfolder and right-click the name of the configuration you wish to edit. From the shortcut menu, select **Edit Projects** (**Figure 5.45**).
2. The **All Projects Editor** will appear (**Figure 5.47**).
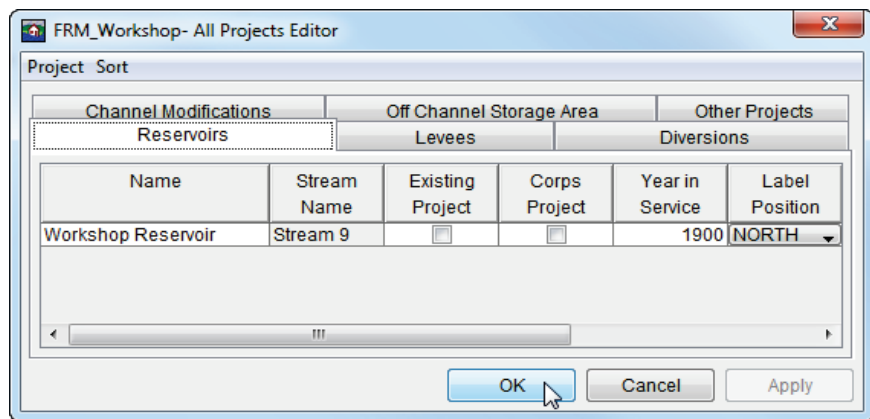
Figure 5.47 The All Projects Editor, used to edit all projects in a configuration.

3. Within the editor, there are tabs for each type of project extant in the configuration. From the editor, users may edit some of the editable properties for each project:

   a. For **Reservoirs**, users may edit reservoir name, project type (Existing Project or Corps Project), service year, label position, and description.

   b. For **Levees**, users may edit levee name, stream bank position, upstream and downstream station, service year, project type (Existing Project or Corps Project), whether or not the levee follows the stream, and description.

   c. For **Diversions**, users may edit diversion name, upstream and downstream station, ending stream name, project type (Existing Project or Corps Project), service year, and description.

   d. For **Channel Modifications**, users may edit channel modification name, starting and ending station, project type (Existing Project or Corps Project), service year, and description.

   e. For **Off Channel Storage Areas**, users may edit off channel storage area name, stream association, project type (Existing Project or Corps Project), service year, label position, and description.

   f. For **Other Projects**, users may edit project name, stream association, stream connection, project type (Existing Project or Corps Project), service year, label position, computation point properties, and description.

4. Click **Apply** to see changes in the **Map Window**. When finished, select **OK** to close the **All Projects Editor**.

5.4   Projects

Once users create configurations, project tools for creating reservoirs, levees, diversions, channel modifications, off channel storage areas, and other projects will appear and users can start adding projects to configurations. **Projects** are man-made structures that are built to conserve water for several purposes. In FIA, the two most important projects are reservoirs and levees. There are several other projects that can be represented in Version 2.1 of FIA, but only reservoirs and levees are used in the calculation of impacts. The following sections describe reservoirs and levees in FIA.

5.4.1   Reservoirs

A **reservoir** is a structure built on a stream to conserve water for uses such as flood damage reduction, hydropower generation, and water supply. The releases from a reservoir can reduce flood damage by reducing flood stages.

5.4.2   Create a Reservoir

To add a reservoir:

Reservoir Tool ▲

1.   You must be in the **Map Window**.
2.   Make sure you have created a configuration to access the project tools. On the map toolbar, engage the **Reservoir Tool**.
3.   Hold down the **CTRL** key and click on the stream alignment to place the upstream end of the reservoir. Release the **CTRL** key; drag the mouse to the downstream end of the reservoir and click.
4.   The **Name New Reservoir** dialog box will open (Figure 5.48). Enter the new **Name** (required) and **Description** (optional). Click **OK**. The **Name New Reservoir** dialog box will close.

Figure 5.48 Name New Reservoir dialog box.

5.   The reservoir will now appear in the **Map Window** as a cyan triangle (reservoir pool), a blue line (storage reach), a gray rectangle (dam), and two computation points that represent the upstream and downstream ends of the reservoir, as shown in Figure 5.49.
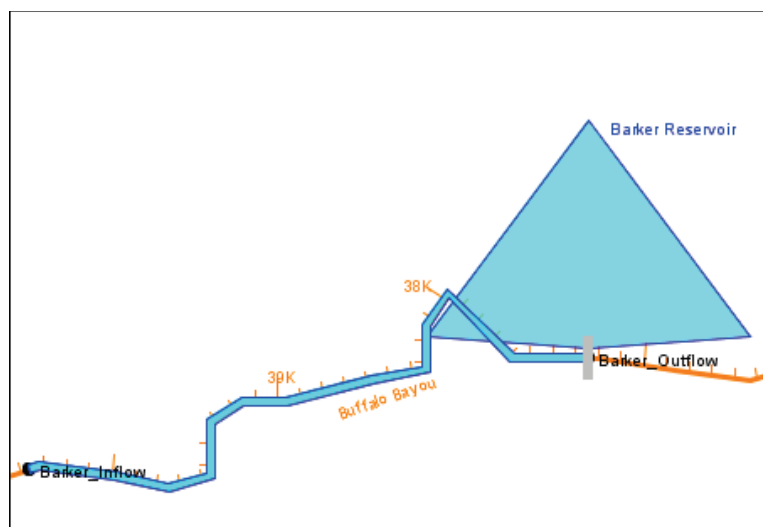
Figure 5.49 Reservoir added to the Map Window.

6. The default representation of the reservoir pool is a triangle; you can edit this triangle if you want a more detailed representation of the pool. With the **Reservoir Tool** pressed, double click on the reservoir. The triangle (Figure 5.50) will now be green with a blue outline color. Also, there are three vertex points (blue), and at the apex of the triangle is the reservoir label, which is represented by a yellow circle.
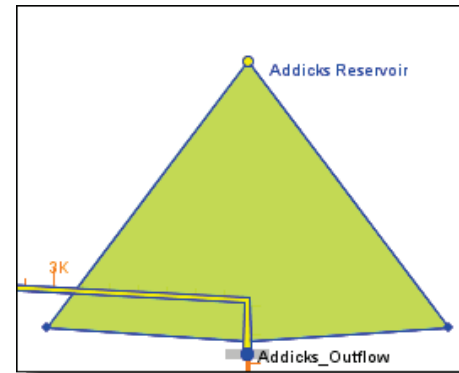
Figure 5.50 Reservoir Pool.

7. By clicking and dragging the yellow circle you can move the reservoir label to an appropriate location. Just like on the stream alignment you can add, move, and delete vertex points (see Section 0, page 5-6). Figure 5.51 illustrates a reservoir pool that has been edited to give a more detailed representation.
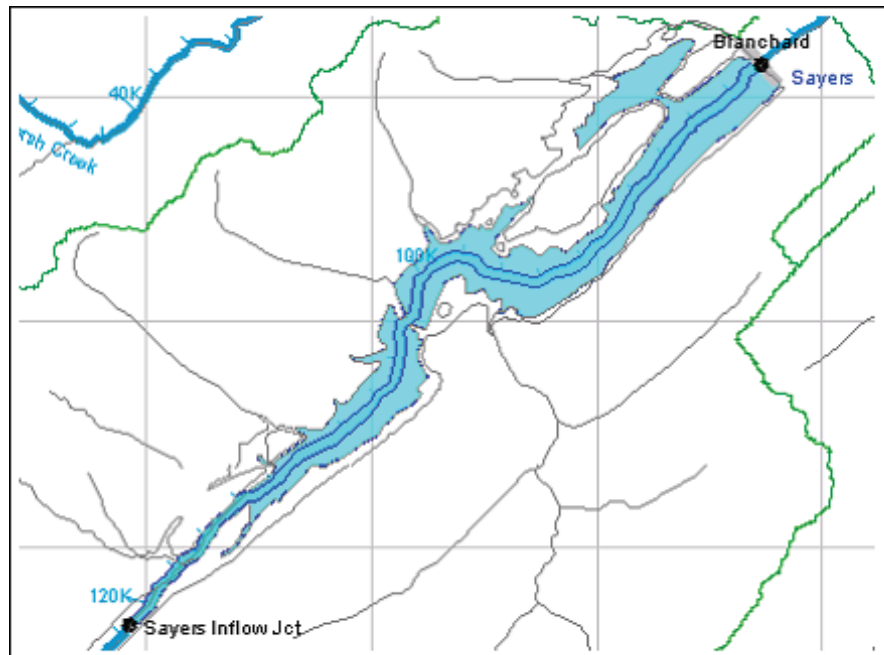
Figure 5.51 Detailed representation of a reservoir pool.

8. When a reservoir is created there are two computation points that are created with default names. It is recommended that you change the names of the two computation points. These two computation points represent the inflow and outflow junctions of the reservoir and should be renamed accordingly.

9. To change the name of a computation point, press the **Computation Point Tool**, right click on one of the computation points. From the shortcut menu, click **Rename Computation Point**; the **Rename**

Computation Point Tool

Computation Point dialog box will open (Figure 5.52). In the **Name** box, enter the new name, and in the **Description** box enter a description.
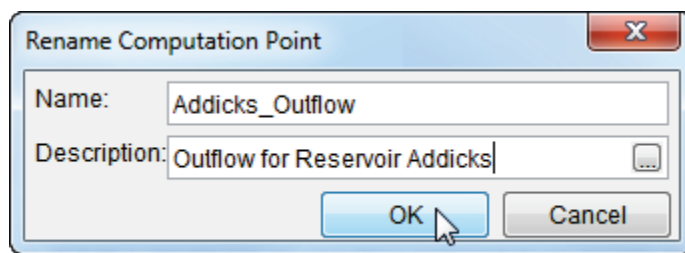
Figure 5.52 Rename Computation Point dialog box.

5.4.3   Edit an Existing Reservoir

After drawing a reservoir in the **Map Window**, you can rename an individual reservoir, remove a reservoir from the active configuration, delete individual reservoirs, create configuration notes for individual reservoirs, and edit individual reservoir properties. All of these tasks are accomplished through the reservoir shortcut menu (Figure 5.53). To access the reservoir element shortcut menu, you must be in the **Map Window**, with the **Reservoir Tool** ▲ engaged.
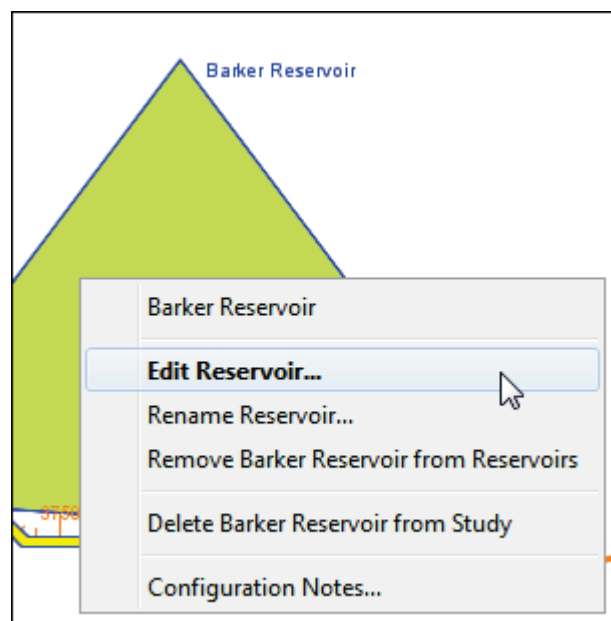
Figure 5.53 Reservoir shortcut menu.

5.4.3.1   Rename a Reservoir

To rename a reservoir:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Reservoir Tool** ▲.
3. Right-click on the reservoir that you want to rename, and click **Rename Reservoir** from the available shortcut menu (Figure 5.53).
4. The **Rename Reservoir** dialog box will open (Figure 5.54). Enter the new **Name** (required) and **Description** (optional) for the reservoir.

5. Click **OK**. A
    warning
    message will
    open asking you
    if you really
    want to rename
    the selected
    reservoir. To
    rename, click
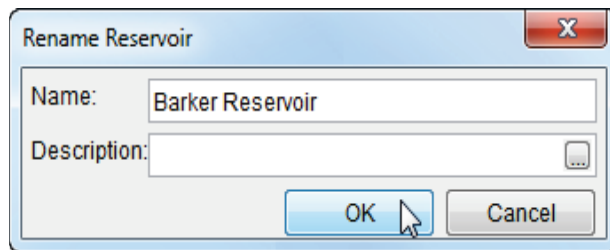
Figure 5.54 Rename Reservoir dialog box.

Yes. The warning message and **Rename Reservoir** dialog boxes
will close, and the reservoir will display the new name.

5.4.3.2   Remove from Configuration

FIA allows you to remove reservoirs from existing configurations. The
reservoir will not be removed from the watershed, only from the selected
configuration. For further information on configurations, see Section 5.3.

To remove a reservoir from a configuration:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Reservoir Tool** ▲.
3. Right-click on the reservoir you want to remove from the active
    configuration, and click **Remove [*Reservoir Name*] Reservoir
    from Reservoirs** from the available shortcut menu (Figure 5.53).
4. A warning message will open asking you if you are sure you want
    to remove the selected reservoir from the current configuration.
    Click **Yes**. The warning message will close, and the reservoir will
    disappear from the **Map Window**.

5.4.3.3   Delete a Reservoir

To delete a reservoir:

1. You must be in the **Map Window**.
2. On the map toolbar, press the **Reservoir Tool**.
3. Right-click on the reservoir you want to delete, and click **Delete
    [*Reservoir Name*] Reservoir from Study** from the available
    shortcut menu (Figure 5.53).
4. A warning message will open asking you if you really want to
    delete the selected reservoir. Click **Yes**. Another warning message
    will open asking you if you wish to delete the computation points
    associated with the reservoir. Click **Yes**. The warning message will
    close, the reservoir and computation points will no longer appear in
    the **Map Window** and both have been removed from the watershed.

5.4.3.4   Add Configuration Notes for a Reservoir

Configuration notes allow you to add information about a reservoir specific to a configuration. Configurations notes are helpful when multiple configurations have been defined. You do not need configuration notes for the default configuration Study. For further information on configurations, see Section 5.3.

To add configuration notes for a reservoir:

1. You must be in the **Map Window**.
2. On the map toolbar, press the **Reservoir Tool**.
3. Right-click on the reservoir to which you want to add configuration notes, and click **Configuration Notes** from the available shortcut menu (Figure 5.53).
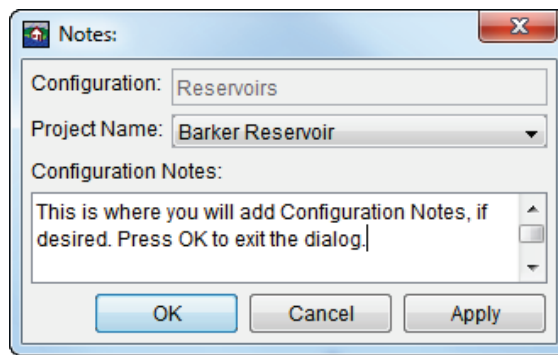
Figure 5.55 Enter Configuration Notes in the Notes dialog box.

4. The **Notes:** dialog box (Figure 5.55) will open. Enter information in the **Configuration Notes** box that describes the selected reservoir for the current configuration.
5. Press **OK** to exit the dialog.

5.4.3.5   Edit Reservoir Properties

In the **Map Window**, you describe the general properties of a reservoir. You can enter a description, whether the reservoir is an existing project, whether the reservoir is a Corps project, and enter the year the reservoir came on-line.

To edit reservoir properties:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Reservoir Tool** ▲.
3. Right-click on the reservoir for which you would like to change properties, and click **Edit Reservoir** from the available shortcut menu (Figure 5.53).
4. The **Reservoir Properties Editor** will open (Figure 5.56). From the **Description** box you can edit or add a description. From the **Reservoir Name** dropdown list, you can also select other reservoirs to edit in your study from this dialog box.
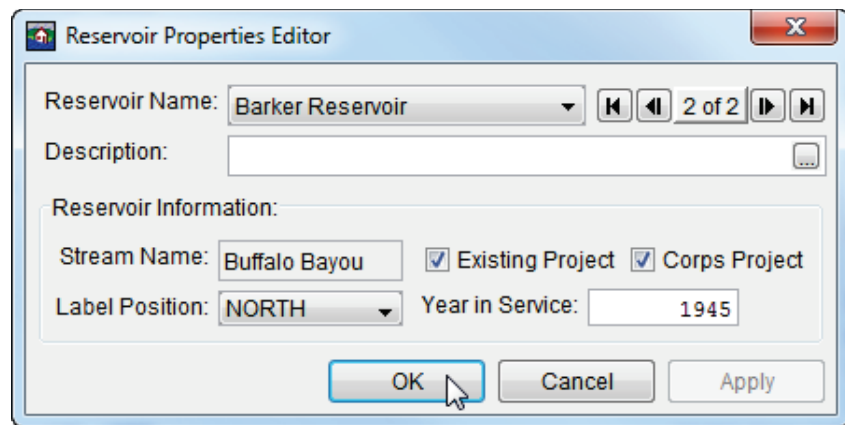
Figure 5.56 Reservoir Properties Editor.

5. If the selected reservoir is an existing project, select the **Existing Project** check box.

6. If the reservoir is a Corps project, select the **Corps Project** check box.

7. If you know what year the reservoir came on-line, enter a year in the **Year in Service** box. This is for information only and not used in the computation of damages.

8. When finished editing, at the bottom of the dialog select **OK**. The dialog will close and your edits will be saved.

5.4.4   Edit Reservoir Drawing Properties

You can override the default colors, line size, and fonts used in drawing a reservoir. Also, you can turn on and off the drawing of the reservoir polygon and the reservoir name.

To edit reservoir drawing properties:

1. Starting in the **Content Pane**, select the **Maps** tab. Under the **Map Layers** folder, locate **Reservoirs** and click (the name will highlight in blue). Right-click and from the shortcut menu select **Properties**.

2. The **Drawing Properties Editor** will open (Figure 5.57).
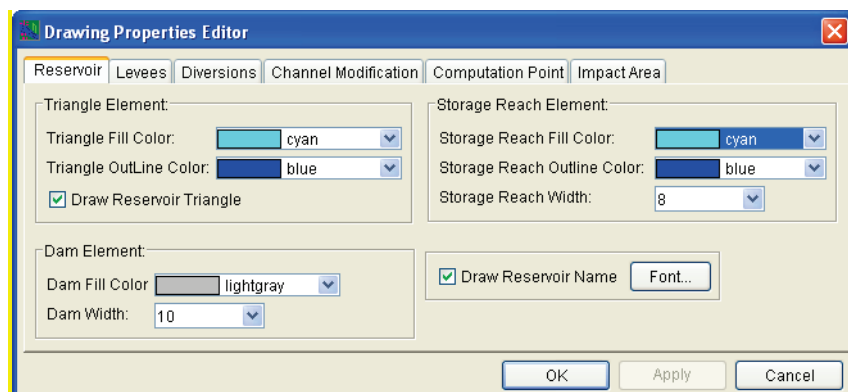
Figure 5.57 Drawing Properties Editor.

3. Click the **Reservoir** tab. To change the default fill and outline color of reservoir polygons (triangle), from the **Triangle Element** box, select a pre-defined color from the **Triangle Fill Color** list and the **Triangle Outline Color** list.

4. To turn off the reservoir polygons in the **Map Window**, clear the **Draw Reservoir Triangle** check box, located in the **Triangle Element** box.

5. To change the default fill and outline color of storage reaches, from the **Storage Reach Element** box, select a pre-defined color from the **Storage Reach Fill Color** list and the **Storage Reach Outline Color** list. To change the width of storage reaches, select a different value from the **Storage Reach Width** list.

5. To change the default fill color of the dam, from the **Dam Element** box, select a pre-defined color from the **Dam Fill Color** list. Change the width of the dam by selecting a different value from the **Dam Width** list.

6. To turn off the reservoir name in the **Map Window**, clear the **Draw Reservoir Name** check box.

7. To change the font, size, and style of the reservoir name, click **Font**. A **Font Chooser** will open (see Figure 5.72).

8. From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, select a different font from the **Type** list. Use the **Size** slider to change the size of the font. As you move the slider, the size changes are shown in the box at the top of the **Font Chooser**.

9. To display any of the changes made in steps 1 - 8 to the **Map Window**, click **Apply** (which will change the **Map Window** but will not close the **Drawing Properties** dialog box) or **OK** (which will change the **Map Window** and will close the **Drawing Properties** dialog box), from the **Drawing Properties** dialog box.

5.4.5   List of all Reservoirs

Users can view a list of all the reservoirs in their study.

To view the list of reservoirs:

1. Starting in the **Study Pane**, under the **Watershed** folder expand the **Project Configurations** subfolder.

2. Right-click on **Reservoirs** and on the shortcut menu point the cursor to **Reports**, then select **List of All Reservoirs** (**Figure 5.58**).

3. The List of All Reservoirs dialog box will open (similar to **Figure 5.42**, page 5-29).

4. The report provides a list of reservoirs that have been input for the study along with various other information about each reservoir.
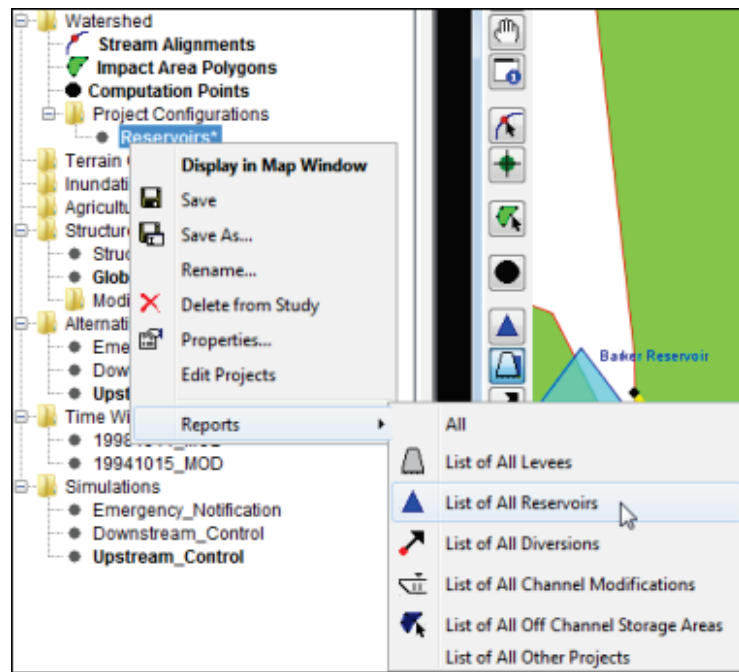
Figure 5.58 Study Pane—Reservoirs shortcut menu.

5. From the **File** menu, you can send the report to a printer, or you can export the report to an ASCII tab delimited file.
6. When done viewing the report, click **Close**. The **List of All Reservoirs** dialog box will close.

5.5   Levees

A **levee** is an earthen barrier built along a stream to provide protection from flooding. A levee can reduce flood damage by preventing flood stages from reaching a potential damage area.

5.5.1   Create a Levee

To add a levee:

Levee	
Tool	

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Levee Tool** .
3. Hold down the **CTRL** key and click on the stream alignment to place the upstream end of the levee. Release the **CTRL** key, drag the mouse to the downstream end of the levee and click.
4. The **Name New Levee** dialog box will open. Enter the new **Name** (required) and **Description** (optional). Click **OK**. The **Name New Levee** dialog box will close.
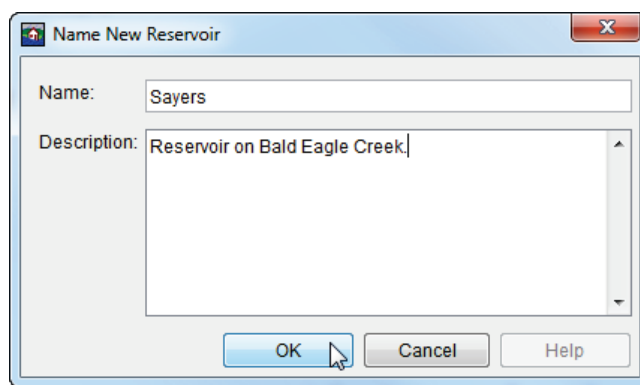5. The levee will now appear in the **Map Window** as a black line, as shown in Figure 5.59.
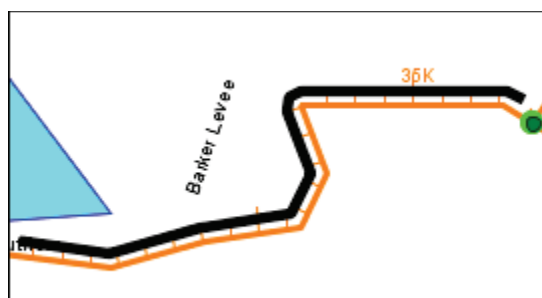
Figure 5.59 New levee displaying in the Map Window.

5.5.2   Edit an Existing Levee

After drawing a levee in the **Map Window**, you can rename an individual levee, remove a levee from the active configuration, delete individual levees, create configuration notes for individual levees, and edit individual levee properties. All of these tasks are accomplished through the levee shortcut menu (Figure 5.60). To access the levee shortcut menu, you must be in the **Map Window** with the **Levee Tool** 🛆 engaged.
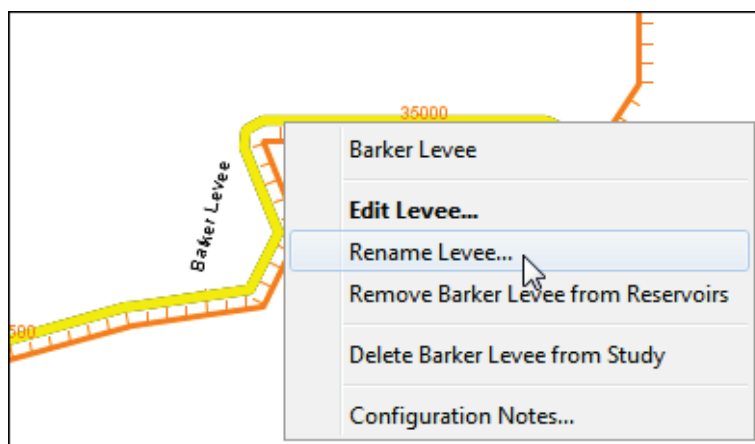
Figure 5.60 Levee shortcut menu.

5.5.2.1   Rename a Levee

To rename a levee:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Levee Tool** 🛆.
3. Right-click on the levee that you want to rename, and click **Rename Levee** from the available shortcut menu (Figure 5.60).
4. The **Rename Levee** dialog box will open. Enter the new **Name** (required) and **Description** (optional) for the levee.
5. Click **OK**. A warning message will open asking you if you really want to rename the selected levee. Click **Yes**. The warning message

and the **Rename** dialog boxes will close, and the levee will display the new name.

5.5.2.2   Remove from Configuration

FIA allows you to remove levees from existing configurations. You cannot remove levees from the default configuration Study. The levee will not been removed from the watershed, only from the selected configuration. For further information on configurations, see Section 5.3.

To remove a levee from a configuration:

1.   You must be in the **Map Window**.
2.   On the map toolbar, press the **Levee Tool**.
3.   Right-click on the levee you want to remove from the active configuration, and click **Remove [*Levee Name*] Levee from Reservoirs** from the available shortcut menu (Figure 5.60).
4.   A warning message will open asking if you are sure you want to remove the selected levee from the current configuration. To remove, click **Yes**. The warning message will close, and the levee will disappear from the **Map Window**.

5.5.2.3   Delete a Levee

To delete a levee:

1.   You must be in the **Map Window**.
2.   On the map toolbar, engage the **Levee Tool** ⌂.
3.   Right-click on the levee you want to delete, and click **Delete [*Levee Name*] Levee from Study** from the available shortcut menu (Figure 5.60).
4.   A warning message will open asking if you really want to delete the selected levee. To delete, click **Yes**. The warning message will close. The levee will no longer appear in the **Map Window**, and it will be permanently deleted.

5.5.2.4   Add Configuration Notes for a Levee

Configuration notes allow you to add information about a levee specific to a configuration. Configurations notes are helpful when multiple configurations have been defined. You do not need configuration notes for the default configuration Study. For further information on configurations, see Section 5.3.

To add configuration notes for a levee:
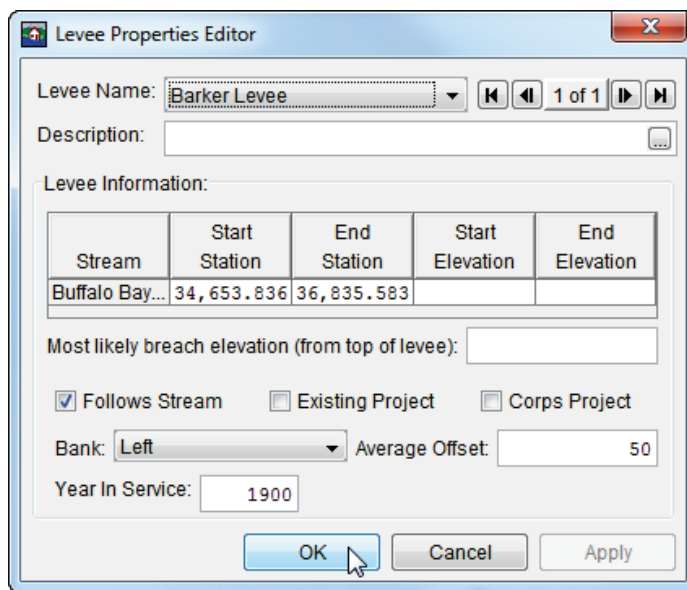
1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Levee Tool** 🔔.
3. Right-click on the levee to which you want to add configuration notes, and click **Configuration Notes** from the available shortcut menu (Figure 5.60).
4. The **Notes:** dialog box will open (see Figure 5.55, page 5-38). Enter information in the **Configuration Notes** box that describes the selected levee for the current configuration.
5. Once you are finished entering, click **OK**. The dialog box will close and configuration notes will be added for your levee.

5.5.2.5   Edit Levee Properties

In the **Map Window**, you are describing the general and aesthetic properties of a levee. The levee properties that you can edit in the **Map Window** include: the description of the levee, whether the levee is an existing project, whether the levee is a Corps project, the year the levee came on-line, what stream the levee is located on, the start and end stations of the levee, adjust the distance (offset) from the stream the levee is placed, and what stream bank the levee is located on.

To edit levee properties:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Levee Tool** 🔔.
3. Right-click on the levee for which you would like to edit properties, and click **Edit Levee** from the available shortcut menu (Figure 5.60).
4. The **Levee Properties Editor** will open (Figure 5.61).

Figure 5.61 Levee Properties Editor.

5. From the **Description** box you can edit or add a description.
6. If the selected levee is an existing project, click the **Existing Project** checkbox.
7. If the levee is a Corps project, select the **Corps Project** checkbox.
8. The table shows the stream the levee is on and the stream stationing for the starting and ending stations of the levee. If you want to change the starting and ending stations of the levee, edit the cells under the **Start Station** and **End Station** columns in the table.
9. By default a levee will follow a stream, so the **Follows Stream** check box will be selected. However, not all levees follow a stream exactly, so if you want to give a more detailed representation of the levee, clear **Follows Stream**. Once this box is cleared, if you double click on the levee with the **Levee Tool** ▣ engaged, vertex points (black circles) appear. Just like on the stream alignment you can add, move, and delete vertex points (see Section 0, page 5-6 for adding/moving vertices). Figure 5.62 shows an edited form of the levee shown in Figure 5.59 (page 5-42).
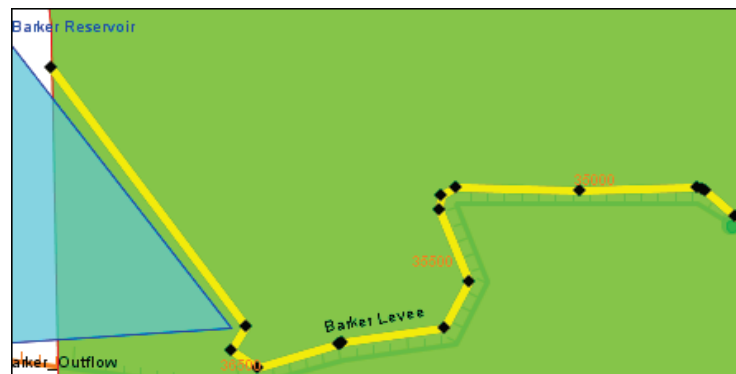
Figure 5.62 Levee that has been manually edited by adding and moving vertices.

10. The program automatically determines which stream bank the levee is located on. If you want to change the stream bank location, from the **Bank** list select the appropriate stream bank.
11. If you know what year the levee came on-line, enter a year in the **Year in Service** box. Service year is for information only and not used to compute damages.
12. When a levee is created, a default-offset distance from the stream alignment is set. The default offset is shown in the **Average Offset** box and is 50. To change the offset, enter a new value in the **Average Offset** box and click **Apply**. The levee on the **Map Window** will move according to the new offset value.

5.5.3   Edit Levee Drawing Properties

You can override the default colors, line size, fonts used in drawing a levee, and also you can turn on and off the levee name. To edit levee drawing properties:

1. You must be in the **Map Window**.
2. From the **Edit** menu, choose **Drawing Properties**. The **Drawing Properties Editor** will open. Click the **Levees** tab (see Figure 5.63).

Figure 5.63 Drawing Properties Editor—Levees tab.

3. To change the default outline color of the levee, you would change the colors in the **Levee Line Color** list.
4. To change the width of the levee, select a value from the **Levee Width** list.
5. To turn off the levee name on the **Map Window**, clear the **Draw Levee Name** check box.
6. To change the font, size, and style of the levee name, click **Font**. A **Font Chooser** (see Figure 7.8, page 7-7) will open.
7. From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, select a different front from the **Type** list. Use the **Size** slider to change the size of the font. As you move the slider, the size changes are shown in the box at the top of the **Font Chooser**.

5.5.4   List of All Levees

FIA allows you to view a comprehensive list of the levees in your watershed.

To view the **List of All Levees**:

1. Starting in the **Study Pane**, under the **Watershed** folder expand the **Project Configurations** subfolder.
2. Right-click on **Reservoirs** and on the shortcut menu point the cursor to **Reports**, then select **List of All Levees** (**Figure 5.58**).
3. The **List of All Levees** dialog box will open (similar to **Figure 5.42**).
4. The report provides a list of levees that have been input for the study along with various other information about each levee.

5. From the **File** menu, you can send the report to a printer, or you can export the report to an ASCII tab delimited file.
6. When done viewing the report, click **Close**. The **List of All Levees** dialog box will close.

5.6 Creating Computation Points

The next step in defining the common schematic is to create your computation points. In time-series-based simulations, computation points are the locations where time-series information is to be exchanged between computer programs. In HEC-FIA a computation point would be the location of the hydrographs (index location).

5.6.1 Create a Computation Point

To add a computation point to the **Map Window**:
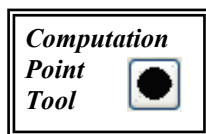
Computation Point Tool

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Computation Point Tool**.
3. Hold down the **CTRL** key and click on the stream alignment to place the computation point.
4. The **Name New Computation Point** dialog box will open (Figure 5.64). Enter the new **Name** (required) and **Description** (optional). Click **OK**, the **Name New Computation Point** dialog box will close.
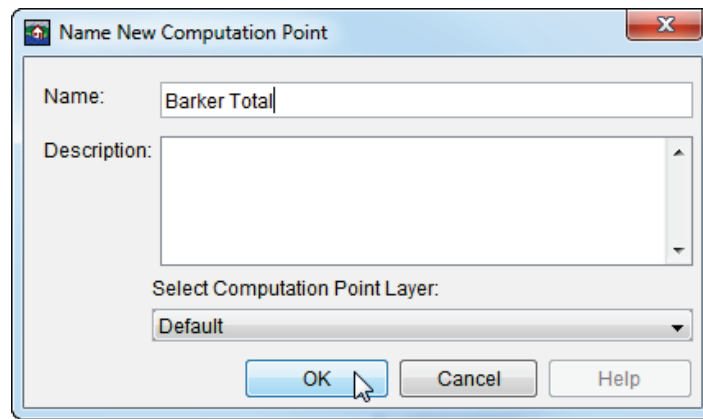
Figure 5.64 Name New Computation Point dialog box.

To add a computation point at a stream junction (see Section 5.1.7, page 5-10):

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Computation Point Tool** .
3. Hold down the **CTRL** key and click on the stream junction to place a computation point.

4. A slightly different **Name New Computation Point** dialog box will open (Figure 5.65). Enter the new **Name** (required) and **Description** (optional).

5. From the **Select the Placement of the Computation Point** list, you determine exactly where the computation point will be placed. You may choose to have the computation point placed on the stream junction, along one of the streams that form the stream junction, or none of the above. For example, in Figure 5.65, the computation point will be placed on the stream junction.

Figure 5.65 Name New Computation Point dialog box (at stream junction).

6. Click **OK**, the **Name New Computation Point** dialog box will close. The computation point will now appear in the **Map Window** as a black circle.

5.6.2   Import Computation Points

To import computation points, you must have uploaded a point shapefile (see Chapter 3, Section 3.1.1, page 3-2) that represents the computation points in your study. This shapefile must contain an attribute that is the stream stationing of each computation point. You should also have either digitized or imported a stream alignment. When it imports, FIA will associate the computation points with existing streams in your study.
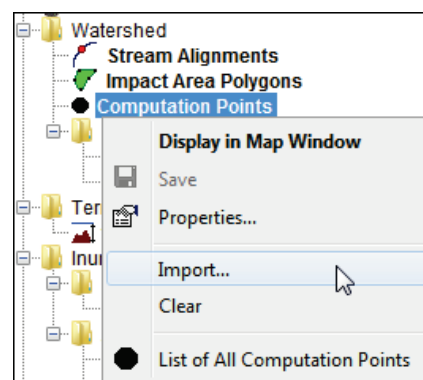
Figure 5.66 Computation Points shortcut menu.

To import computation points:

1. Starting in the **Study Pane**, under the **Watershed** folder of the **Study Tree** right-click **Computation Points**.

2. From the shortcut menu (Figure 5.66), select **Import**.

3. The **Import Computation Points** dialog box will open (Figure 5.67).
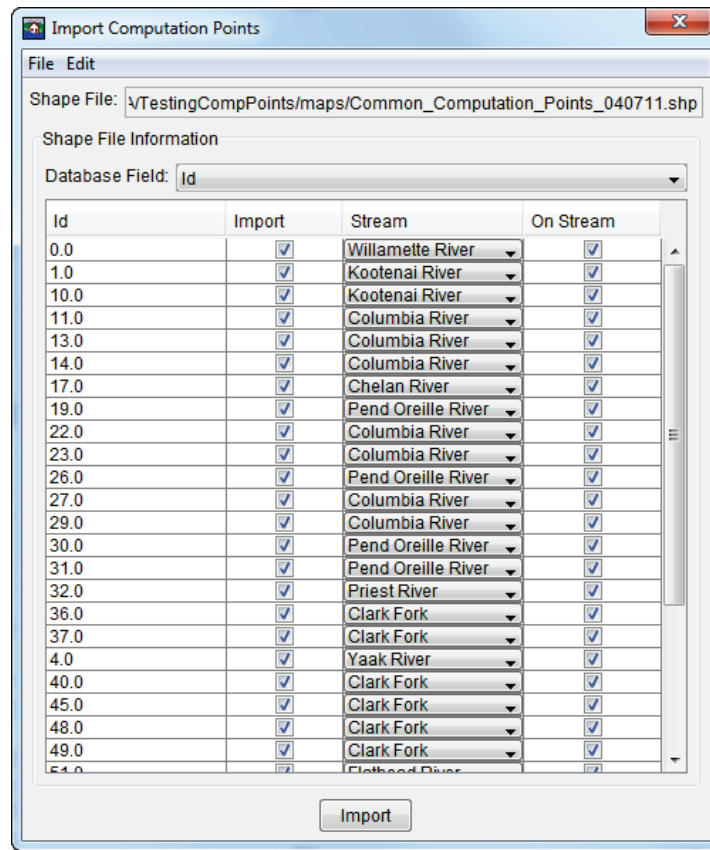
Figure 5.67 Import Computation Points dialog box.

4. The leftmost field (i.e. *Id*) shows the shapefile attribute for stream stationing for the shapefile *Common_Computation_Points_040711.shp*.
5. The checked boxes under the **Import** column represent the computation points that will be imported. All boxes will be checked by default. Uncheck the box(es) you do not want to import.
6. FIA automatically associates the computation points with their corresponding streams. To reassign a computation point to a different stream, select the correct stream from the dropdown list under the **Stream** column.
7. A checked box in the **On Stream** column will import that computation point on the stream. If you would like to import a computation point outside of a stream, uncheck the box in the **On Stream** column.
8. Once all import parameters are set, at the bottom of the dialog click **Import**. A message box will appear, notifying you how many computation points were successfully imported (Figure 5.68) and your computation points will appear in the **Map Window**.
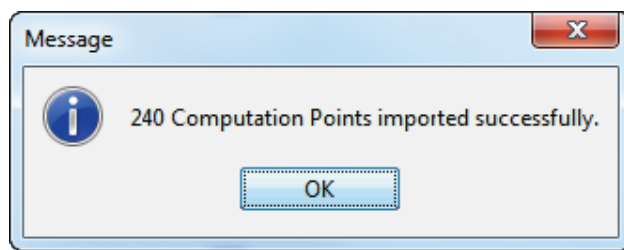
Figure 5.68 Message box.

5.6.3   Edit a Computation Point

After creating a computation point in the **Map Window**, you can rename an individual computation point, delete individual computation points, and edit individual computation point properties. All of these tasks are accomplished through the computation point shortcut menu (Figure 5.69). To access the computation point shortcut menu, you must be in the **Map Window** with the **Computation Point Tool** ⬤ engaged.
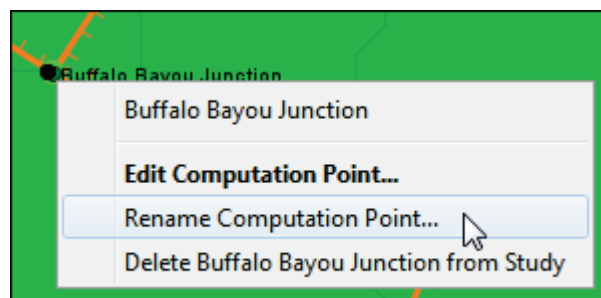
Figure 5.69 Computation Point shortcut menu.

5.6.3.1   Rename a Computation Point

To rename a computation point:

1. With the **Computation Point Tool** ⬤ engaged, right-click on the computation point you want to rename, and click **Rename Computation Point** from the available shortcut menu (Figure 5.69).
2. The **Rename Computation Point** dialog box will open. Enter a new **Name** (required) and a short **Description** (optional) for the common computation point.
3. Click **OK**. A warning message will open asking the user to confirm renaming of the computation point Click **Yes**. The warning message and **Rename Computation Point** dialog boxes will close, and the computation point will display the new name.

5.6.3.2   Delete a Computation Point

Computation points created with a project (i.e., reservoir inflow/outflow computation points) or an impact area cannot be deleted. These

computation points are deleted when the project or impact area is deleted. Any other computation point can be deleted from the shortcut menu in **Figure 5.69**.

To delete a computation point:

1. With the **Computation Point Tool** ⬤ engaged, right-click on the computation point you want to delete, and click **Delete [*Computation Point Name*] from Study** from the available shortcut menu (Figure 5.69).
2. A warning message will open asking the user to confirm deletion of the computation point. Click **Yes** to delete. The warning message will close and the computation point will no longer appear in the **Map Window**.

5.6.3.3   Edit Computation Point Properties

To edit computation point properties:

1. Right-click on the computation point whose properties you would like to change, and click **Edit Computation Point** from the available shortcut menu (Figure 5.69).
2. The **Computation Point Editor** will open (Figure 5.70). From the **Description** box you can edit or add a description
3   By default FIA automatically decides which stream the computation point is located on. The **Stream Name** box provides you with this information and is not editable.

Figure 5.70 Computation Point Properties Editor.

4. FIA automatically generates the stream station for the computation point. If you want to change the stream station, in the **Stream Station** box enter a new value.
5. The **Label Position** list allows you to place the computation label on the **Map Window**.
6. When the computation point is associated with a stream element, its location is displayed by the northing and easting coordinates. These values are not editable.

7. By default, a computation point is placed on a stream in the stream alignment. If you wish to have a computation point that is not on a stream, clear the **Snap to Steam Element** check box.

8. Once you have cleared the **Snap to Stream Element**, you can edit the easting and northing coordinates of the common computation point, and you can move the common computation point off the stream alignment.

9. Once you are finished editing computation point properties, click **OK**. The dialog box will close.

5.6.4 Move a Computation Point

To move a computation point:

1. You must be in the **Map Window**.
2. On the map toolbar, engage the **Computation Point Tool** ⬤.
3. Click and drag the computation point along the stream to a new location.
4. If you want a computation point to exist outside of your stream alignment, see the previous section **Edit Computation Point Properties** on page 5-51.

5.6.5 Edit Computation Point Drawing Properties

You can override the default colors, line size, fonts used in drawing a computation point, and also you can turn on and off the computation point name.

To edit computation point drawing properties:

1. You must be in the **Map Window**.
2. From the **Edit** menu, click **Drawing Properties**. The **Drawing Properties Editor** will open. Click the **Computation Point** tab (Figure 5.71).
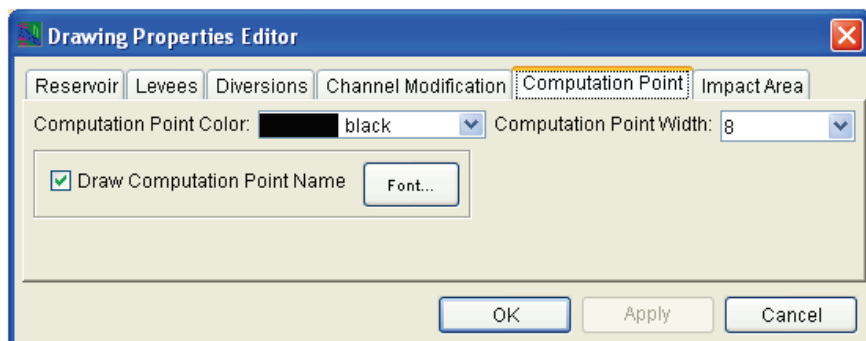
Figure 5.71 Drawing Properties Editor.

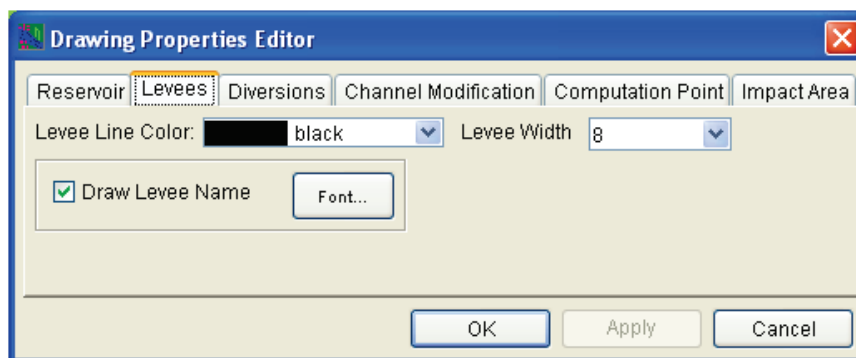3. To change the default color of the computation point, select a different color from the **Computation Point Color** list.

4. To change the default width of the computation point, select a different value from the **Computation Point Width** list.
5. To turn off the computation point name in the **Map Window**, clear the **Draw Computation Point Name** check box.
6. To change the font, size, and style of the computation point name, click **Name Font**. A **Font Chooser** will open (Figure 5.72).
7. From the **Font Chooser** you can change the font type, the size of the font, and whether you want the name to be in bold or italics. To change the font type, select a different font from the **Type** list. Use the **Size** slider to change the size of the font. As you move the slider, the size changes are shown in the box at the top of the **Font Chooser**.
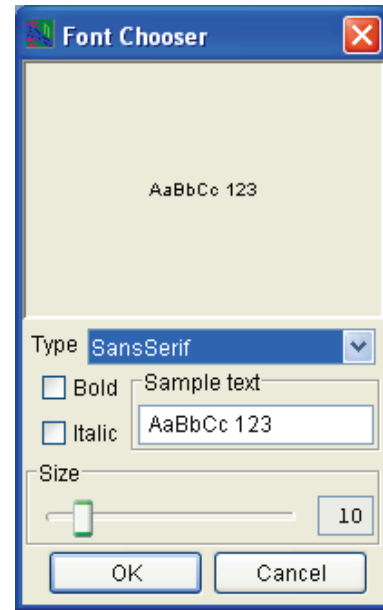
Figure 5.72 Font Chooser.

5.6.6   List of Computation Points

FIA allows you to view a comprehensive list of the computation points in your watershed. To view the **List of Computation Points**:

1. Starting in the **Study Pane**, under the **Watershed** folder expand the **Project Configurations** subfolder.
2. Right-click on **Computation Points** and on the shortcut menu point the cursor to **Reports**, then select **List of All Computation Points** (see **Figure 5.58**).The **List of All Computation Points** dialog box will open (similar to **Figure 5.42**, page 5-29).
3. The report provides a list of computation points that have been input for the watershed, along with the computation point's description, stream association, project association (if any), and stream station.
4. From the **File** menu, you can send the report to a printer, or you can export the report to an ASCII tab delimited file.
5. Click **Close**, the will close the **List of Computation Points** dialog box.

Chapter 6

Gridded Data

HEC-FIA allows users to add gridded data to compute economic damages and life loss. This section will cover two categories of gridded data: **Terrain Data** and **Inundation Data**. Inundation data in HEC-FIA consists of **Depth Grids**, **Arrival Time Grids**, and **Duration Grids**.

If the inundation grid has been split into more than one grid (i.e. it has been **tiled**) then the inundation data will need to be loaded in as a **Group** in HEC-FIA. **Tiled** grids are multiple grids that combine to act as a single contiguous grid. RAS Mapper has the ability to work with tiled data so that output can come in tiled format. Terrain, depth, duration, and arrival time grids can all be imported in tiled format with the **Group** option.

6.1   Terrain Data

Terrain Data is required in HEC-FIA to assign elevations to structures. HEC-FIA accepts terrain data in ASCII or binary (.flt) digital elevation grid format. For details on preparing terrain data for use in FIA, see Appendix B. If FIA is going to be used to generate a structure inventory, or to update elevations of an imported inventory, a terrain grid must be supplied in the form of a digital elevation grid.

6.1.1   Add Terrain Grid

To load a terrain grid in HEC-FIA:

1.   In the **Study Pane**, right-click on the **Terrain Grids** folder. From the shortcut menu select **Add Terrain Model** (Figure 6.1).
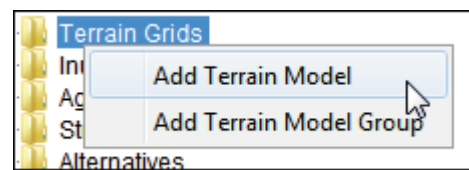
Figure 6.1 Terrain Grids folder shortcut menu.

2. The **Open Browser** will open. Browse to the location of the binary or ASCII files you want to associate with terrain data.

3. From the **Files of type** list, select the appropriate file extension for the data you wish to use. In Figure 6.2, ESRI FLT *(*.flt)* has been selected. Select the file that represents your terrain model and click open.
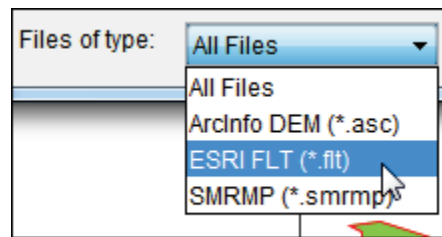
Figure 6.2 Files of type list.

4. The **Open** browser will close and a message box will appear (Figure 6.3), asking whether you want to convert the binary file to an HEC-FIA format, **SMRMP** (Segmented Multi-Resolution Multi-Parameter). This conversion is not required, but is recommended. HEC-FIA will run more efficiently with grids that are converted to the SMRMP format, especially large grids.
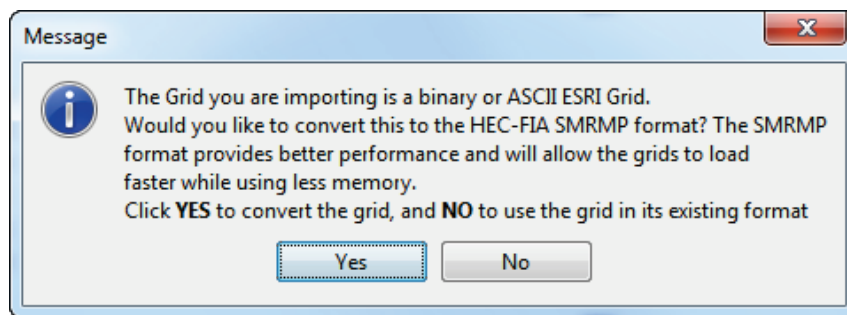
Figure 6.3 Message box.

5. Click **Yes**. The message box will close and a progress dialog box will open, displaying the progress of creating the SMRMP file (i.e., *rastert_ext_osa1.smrmp)*. This could take several minutes depending on the size of the grid.

6. When the conversion is finished, the progress dialog box will close. The terrain data will now display in the map window and the name of the SMRMP file will appear under the **Terrain Data** folder on the **Study Pane**. The SMRMP file will be created in the **maps** folder of the study directory.

6.1.2   Remove Terrain Grid

To remove a terrain grid from your study:

1. Starting in the **Study Pane**, right click on the filename of the map you wish to remove. From the shortcut menu, select **Remove From Study** (Figure 6.4).
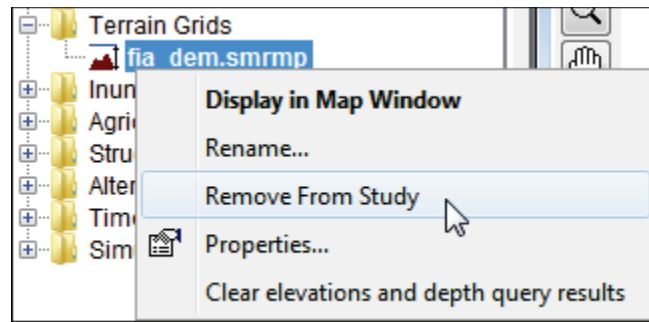
Figure 6.4 Shortcut menu for terrain grids.

2. The terrain grid will now be removed from your study. However, the SMRMP file created by HEC-FIA in the **maps** folder of the study directory will still remain.

3. If you wish to delete this file, locate it in the **maps** folder of the study directory and delete it manually (see Figure 6.5).
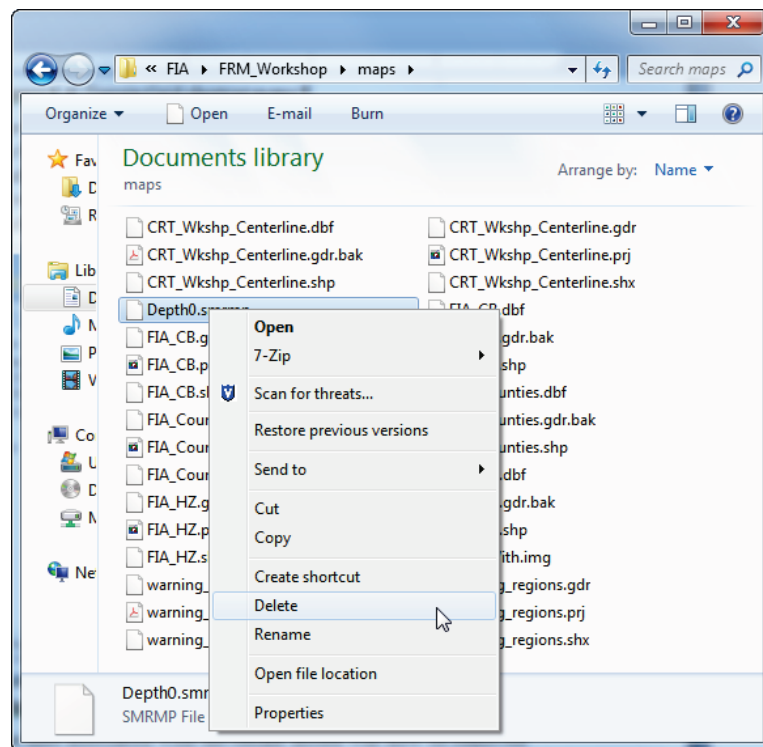
Figure 6.5 Manually deleting a terrain grid (maps folder).

4. The files associated with the terrain model will now be removed from your hard drive.

6.1.3 Rename Terrain Grid

From the **Terrain Grid** shortcut shown in Figure 6.4 (page 6-3), users can also rename a terrain grid.

To rename a terrain grid in HEC-FIA:

1. Starting in the **Study Pane**, under the **Terrain Grids** folder locate the name of the terrain grid you wish to rename.
2. From the shortcut menu, select **Rename.** (see option in Figure 6.4, page 6-3).
3. The **Rename Terrain Grids** dialog box will appear. Enter a new name for your time window and a short description.
4. Another dialog box will appear, prompting the user to confirm renaming of the file (Figure 6.6).
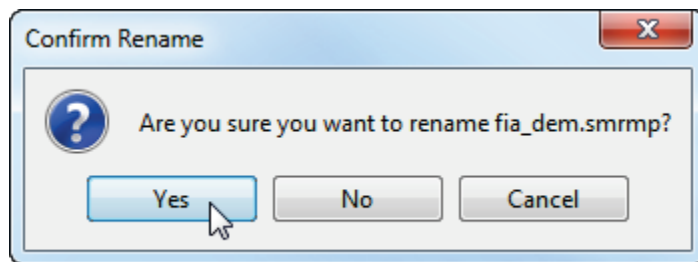
Figure 6.6 Confirm Rename dialog box.

5. Select **No** to cancel renaming and return to your study. Select **Cancel** to return to the **Rename Terrain Grids** dialog and make a change in name or description. Select **Yes** to rename.
6. Your newly-renamed terrain grid will appear under the **Terrain Grids** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and your newly renamed terrain grid, from the toolbar click 💾.

6.1.4   Terrain Grid Groups

If your digital terrain model is composed of several tiled grids, they will need to be uploaded as a **Terrain Grid Group**. In HEC-FIA, **Groups** work by creating and naming a group and adding individual grids to the group.

HEC-FIA will run more efficiently with grids that are converted to the SMRMP format, especially large grids. If your tiled terrain model is not already in SMRMP format, multiple files can be batch converted using this process:

1. From the **Data** menu, select **Convert Grids** (Figure 6.7).
2. An **Open** browser will appear. Browse to the folder where the group of inundation grids is located. Click to select the first grid and then hold shift and click to select the rest of the grids.
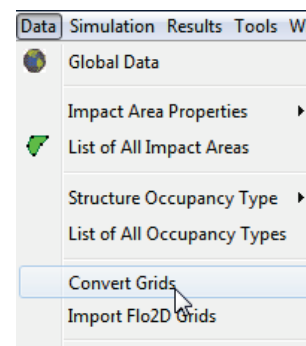
Figure 6.7 Data menu.

3. Once all of the grids have been highlighted, click on the **Select** button. The **Open** browser will close and a status bar (Figure 6.8) will open letting the user know the status of the conversions.
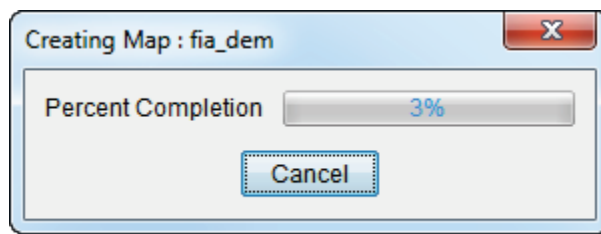
Figure 6.8 Progress dialog box for grid conversion.

4. When batch converting from .flt to .smrmp, the .smrmp files are created within the same folder as the .flt files. If that location is different than the **maps** folder of your study directory, the .smrmp files must be moved to the maps folder after conversion and prior to adding them to the study. This will ensure that the model is portable.

Next, create the **Terrain Grid Group**:

5. After all of the grids have been converted, right click on the **Terrain Grids** folder in the **Study Tree** and from the shortcut menu (Figure 6.1) select **Add Terrain Model Group**.
6. The **Create New Terrain Grids** dialog box (Figure 6.9) will appear. Enter **Name** and **Description** for the Terrain Model Group. FIA creates the new group in the study **Directory** by default; enter or browse ⬚ to a new directory if desired.
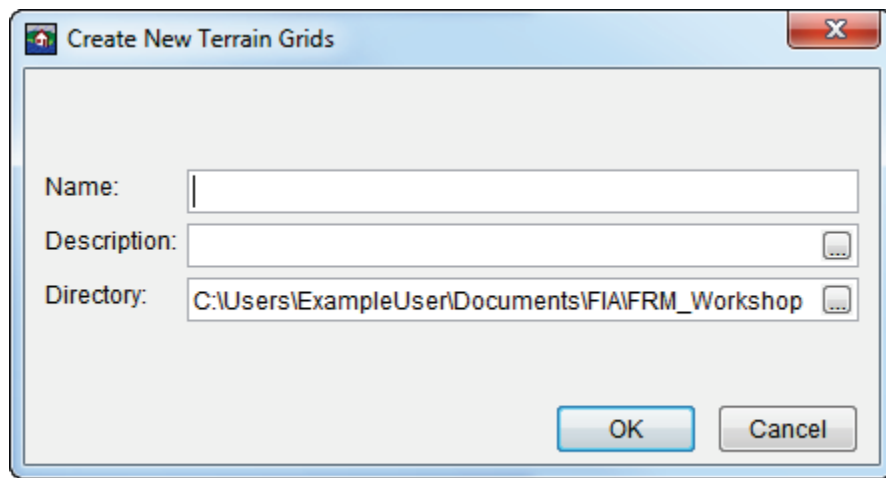
Figure 6.9 Create New Terrain Grids dialog box.

6. The new group will appear under the **Terrain Grids** folder in the **Study Pane**. To add the tiled depth grids to the group, right click on the newly created group and from the shortcut menu (Figure 6.1**0**) select **Add Terrain Model**. An open dialog box will appear.

Browse to the folder where the grouped depth grids are located, select one, and click **Open**.
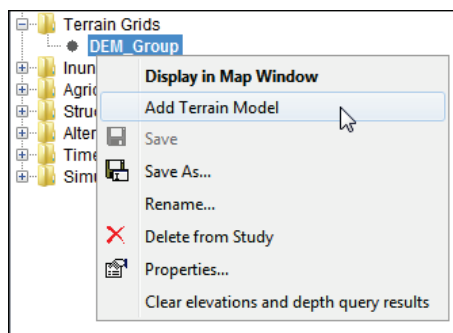
Figure 6.10 Shortcut menu for terrain model groups.

7. The name of the grid you have added to the group will appear under the group's subfolder. Repeat step 5 for all of the other tiled terrain grids in the group.

6.1.5 Delete a Terrain Grid Group

Using the shortcut menu shown in Figure 6.10, users can also delete a terrain grid group:

1. Starting in the **Study Pane**, under the **Depth Grids** folder locate the name of the inundation grid group you wish to delete.
2. From the shortcut menu, select **Delete from Study** (see option in Figure 6.14).
3. A dialog box will appear prompting the user to confirm deletion. To delete, select **Yes**.
4. The terrain grid group will now be removed from your study and your hard drive.

Users can also remove individual terrain grids from a terrain grid group by right-clicking a terrain grid and selecting **Remove from Study** from the shortcut menu (see Figure 6.4).

6.2 Inundation Data Grids

Inundation data is required by HEC-FIA for computing economic and life safety consequences. For a typical rapid consequence assessment, inundation data is supplied in the form of a **Depth Grid** that represents the maximum inundated area along with arrival time information either by supplying hydrographs at each cross-section and storage area location or **Arrival Time Grids**. If hydrographs are used to supply the arrival time information, then only the maximum inundation **Depth Grid** for each scenario needs to be added under the **Inundation Data** folder. (The hydrograph information is entered later when creating a simulation.) If life loss and agricultural damages are going to be assessed with **Duration**

grids and **Arrival Time Grids** then depth, duration, and arrival time grids will all need to be added under the **Inundation Data** folder. When the user is evaluating events, coordination with the hydraulic engineer is required to represent changes in alternatives that are structural. That would include many projects that are typical in floodplain mitigation such as levee setbacks, levee removal, levee raising, storage basins, channel modifications, and many more structural measures. These changes would be represented in the Hydraulic model and therefore be present in the various grids that describe the events.

All three **Inundation Grids** and **Inundation Grid Groups** are added and removed by the same process.

6.2.1   Add an Inundation Data Grid

To load a single **Depth Grid**, **Arrival Time Grid**, or **Duration Grid** in HEC-FIA:

1. Starting in the **Study Tree**, right-click on **Inundation Data**. From the shortcut menu (Figure 6.11) select **Add Depth Grid**, **Add Arrival Time Grid**, or **Add Duration Group** depending on grid type.

2. The **Open** browser will appear. Browse to the location of the binary

Figure 6.11 Inundation Data folder shortcut menu.

(.flt) or ASCII format file (.asc) that contains inundation data.
3. From the **Files of type** list (Figure 6.2), select the file extension for the data you wish to use, then click the appropriate file and click **Select**.
4. The **Open** browser will close and a message box will appear (see Figure 6.3, page 6-3) asking whether you want to convert the binary file to an HEC-FIA format, **SMRMP** (Segmented Multi-Resolution Multi-Parameter). This conversion is not required, but is recommended. HEC-FIA will run more efficiently with grids that are converted to the SMRMP format, especially large grids.
5. Click **Yes**. The message box will close and a progress dialog box will open, displaying the progress of creating the SMRMP file. This could take several minutes depending on the size of the grid.
6. When the conversion is finished, the progress dialog box will close. The inundation grid will now display in the map window and the name of the uploaded grid will appear under the **Inundation Data**

folder of the **Study Pane** with an asterisk (*****) indicating that it is unsaved.

7. To save your study and your new inundation grid, from the toolbar click 💾.

6.2.2   Remove an Inundation Data Grid

To remove a **Depth Grid**, **Arrival Time Grid**, or **Duration Grid** from your study, right click on the filename of the grid you want to remove and follow the instructions in Section 6.1.2 for removal and deletion of gridded data.

6.2.3   Rename an Inundation Grid

To rename a **Depth Grid**, **Arrival Time Grid**, or **Duration Grid** in HEC-FIA:

1. Starting in the **Study Pane**, under the **Inundation Data** folder locate the name of the inundation data grid you wish to rename.
2. From the shortcut menu, select **Rename** (see Figure 6.12).
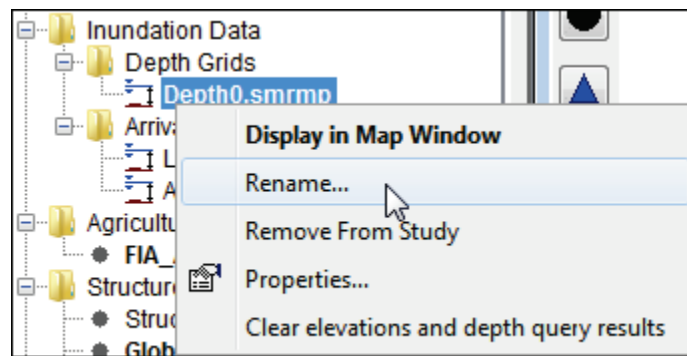
Figure 6.12 Shortcut menu for inundation data grids.

3. A dialog box will appear. Enter a new name for your inundation data grid and a short description.
4. Another dialog box will appear, prompting the user to confirm renaming of the file
5. Select **Yes** to rename.
6. Your newly-renamed inundation data grid will appear under its appropriate subfolder of the **Inundation Data** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and your newly renamed grid, from the toolbar click 🖫 .

6.2.4   Add an Inundation Data Grid Group

The following example will show how to import a depth grid group. **The same process is used for importing Depth, Arrival Time, and Duration Grid Groups.**

HEC-FIA will run more efficiently with grids that are converted to the SMRMP format, especially large grids. If your tiled inundation data is not already in SMRMP format, multiple files can be batch converted using this process:

1. From the **Data** menu (Figure 6.7), select **Convert Grids**.
2. An **Open** browser will appear. Browse to the folder where the tiled inundation grids are located. Click to select the first grid and then hold shift and click to select the rest of the grids.
3. Once all of the grids have been highlighted, click on the **Select** button. The **Open** browser will close and a status bar will open letting the user know the status of the conversions.
4. **The batch convert process does not automatically create the SMRMP grids in the correct location**. You must move them manually to the **maps** folder within the study directory so that the FIA model you create will be portable.

Next, create the **Inundation Data Group**:

5. After all of the grids have been converted, right click on the **Inundation Data** folder in the **Study Tree** and select **Add Depth Group** (see Figure 6.11). The **Create New Inundation Data** dialog box will appear (Figure 6.13). Enter a **Name** and **Description** for your inundation data group. FIA creates the new group in the study **Directory** by default; enter or browse [...] to a new directory if desired.
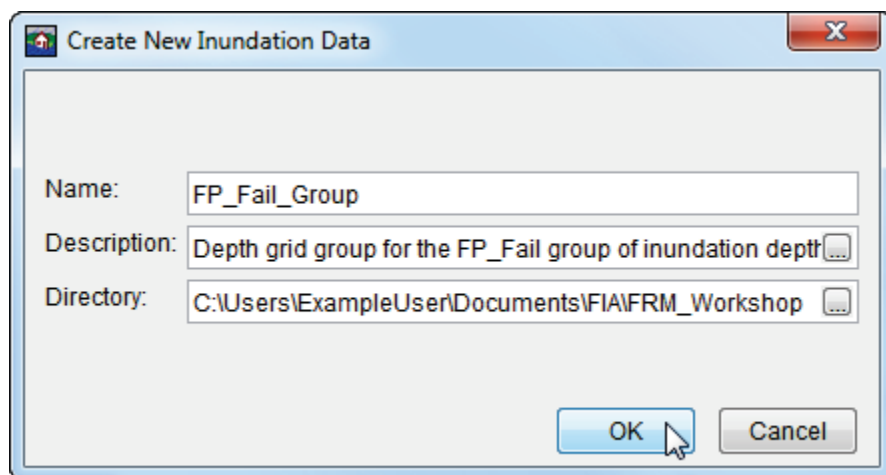
Figure 6.13 Create New Inundation Data dialog box.

5. The group will appear under the **Depth Grids** folder in the **Study Pane**. To add the tiled depth grids to the group, right click on the newly created group and select **Add Depth Grid** (Figure 6.14). An open dialog box will appear. Browse to the folder where the grouped depth grids are located, select one, and click **Open**.
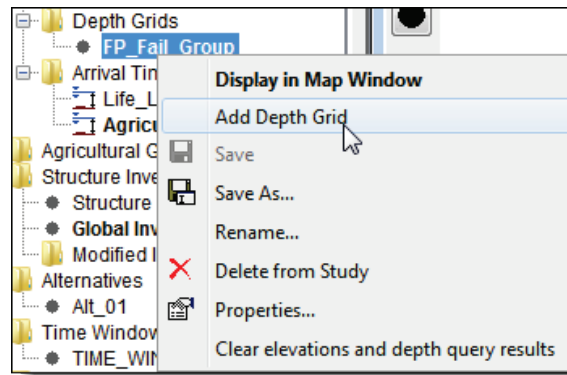
Figure 6.14 Depth Grid Group shortcut menu.

6. The name of the grid you have added to the group will appear under the group's subfolder. Repeat step 5 for all of the other tiled depth grids.

Remember, this process can be done for all tiled inundation data including **Depth, Arrival Time** and **Duration Grids**.

6.2.5 Remove an Inundation Data Grid Group

Using the shortcut menu shown in Figure 6.14, users can also delete an inundation data grid group.

To delete an inundation grid group from a study:

1. Starting in the **Study Pane**, under the **Depth Grids** folder locate the name of the inundation grid group you wish to delete.
2. From the shortcut menu, select **Delete from Study** (see option in Figure 6.14).
3. A dialog box will appear prompting the user to confirm deletion. To delete, select **Yes**.
4. The inundation data grid group will now be removed from your study and your hard drive.

Users can also remove individual inundation grids from an inundation grid group by right-clicking on a grid within the group and selecting **Remove from Study** from the shortcut menu (Figure 6.12).

6.3   Grid Query Tool

Once terrain or inundation data grids have been added to a study, users the **Grid Query Tool** will become functional.

Click on a point of an inundation or terrain grid in the **Map Window** to query the grid (Figure 6.1**5**). For an inundation grid, a click will display depth values. For an arrival time grid, a click will display the computed arrival time of flood waters during an event for the point at which you clicked. For a duration grid, it will display values of the duration of water during a flood event for the cell where you click. For a digital terrain model, the tool queries elevation values.
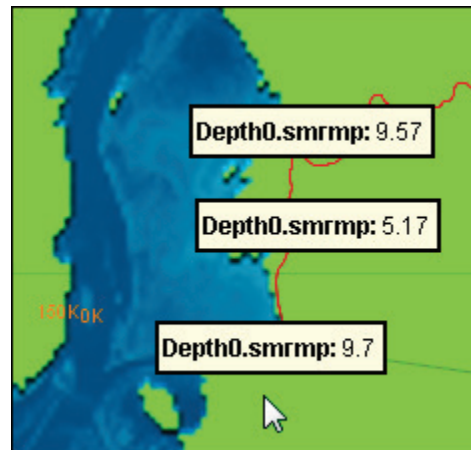
Figure 6.15 Using the Grid Query Tool to query inundation depth values.

To clear the queries, right-click on a terrain or inundation grid and select **Clear elevations and depth query results** or zoom in or out in the **Map Window**.

Chapter 7

Agricultural Data

In order to compute agricultural damage, users add gridded agricultural data that covers the maximum extent of flood waters for the flood events being modeled. The data set is paired with a table in which users define the crop types and values, seasonal data, and damage functions used for each crops. FIA will then use this data in a simulation to compute the monetary damage and area flooded during a flood event. This chapter describes how to add and define an agricultural data set to an FIA study.

7.1   Import Agricultural Data

The geographic information system (GIS) pre-processing that is required is described in Appendix B of this manual. FIA currently accepts agricultural data only in the form of NASS CDL grids.

7.1.1   Import Agricultural Data from NASS CDL Grid

The Cropland Data Layer (CDL) is a geospatial crop-specific digital data layer used in GIS applications. The CDL is provided as a GeoTIFF (.*tif*) by the National Agricultural Statistics Service (NASS) of the United States Department of Agriculture (USDA).

The CDL can be downloaded from the NASS website at: http://www.nass.usda.gov/research/Cropland/SARS1a.htm. For preparation of NASS CDLs, see Appendix B.

To import the NASS CDL grid into HEC-FIA:

1. From the **Study Pane**, right click on the **Agricultural Grids** folder. From the shortcut menu (Figure 7.1), choose **Generate From→ Load NASS CDL Grid**.
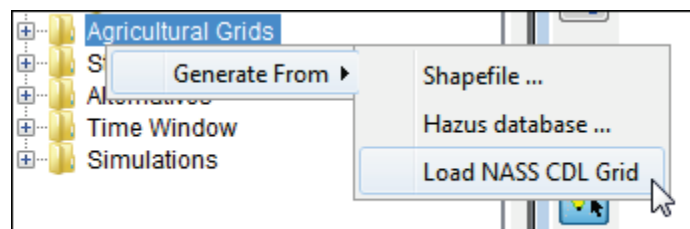
Figure 7.1 Agricultural Grids folder shortcut menu.

2. The **Generate Crop Data From NASS CDL** dialog box will appear (Figure 7.2).
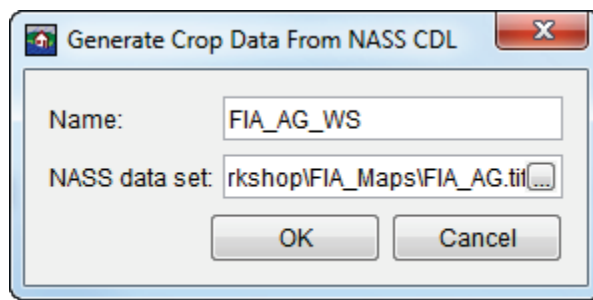
Figure 7.2 Generate Crop Data from NASS CDL dialog.

3. In the **Name** box, enter a name for the imported agricultural data. In the **NASS data set** box click the browse button [...] to select the NASS CDL file (*FIA_AG.tif*) you would like to upload.

4. Click **OK**. The dialog will close and the new agricultural grid will appear under under the **Agricultural Grids** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved (Figure 7.3).

5. To save your study and your new agricultural grid, from the toolbar click [💾].
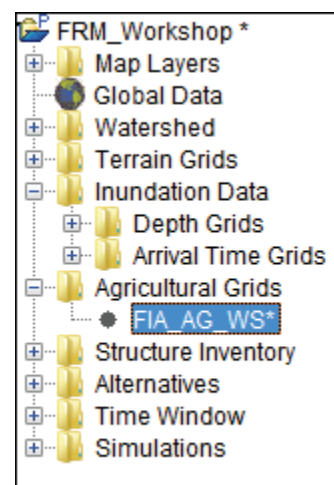
Figure 7.3 Newly added agricultural grid (Study Pane).

Editing Extents

If a GeoTIFF has been created in ArcGIS with an associated world file (.tfw), there should not be a need to edit the extents. Proper spatial reference data is contained in the world file (see Appendix B, Section B.10).

Without a world file, when HEC-FIA pulls the spatial reference data from the GeoTIFF (.tif) file, it will not convert its extents to feet. Thus, if the study is in the Albers Conical Equal Area (NAD_1983_Albers) projected coordinate system, the agricultural data will most likely display incorrectly in the **Map Window** (Figure 7.4).
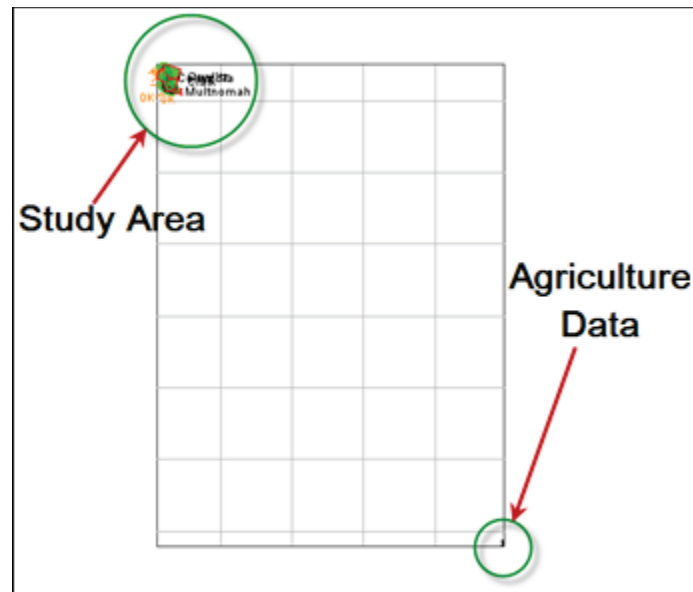
Figure 7.4 Incorrect Extents (Map Window).

If the agricultural data is not correctly mapped with the rest of the study area the extents may need to be corrected.

To edit the extents of a NASS CDL .tif file:

*If the agricultural data is not correctly mapped with the rest of the study area **the extents may need to be corrected***

1.   Open **Windows Explorer** and browse to the HEC-FIA study folder. Within the HEC-FIA study folder, open the folder called **ag** and right click on the file ending in .img (i.e., *westfork_NASS_ag.img*) that is associated with the incorrect agricultural data (Figure 7.6). In the shortcut menu select **Open With** and then if Notepad is not available select **Choose Program**.
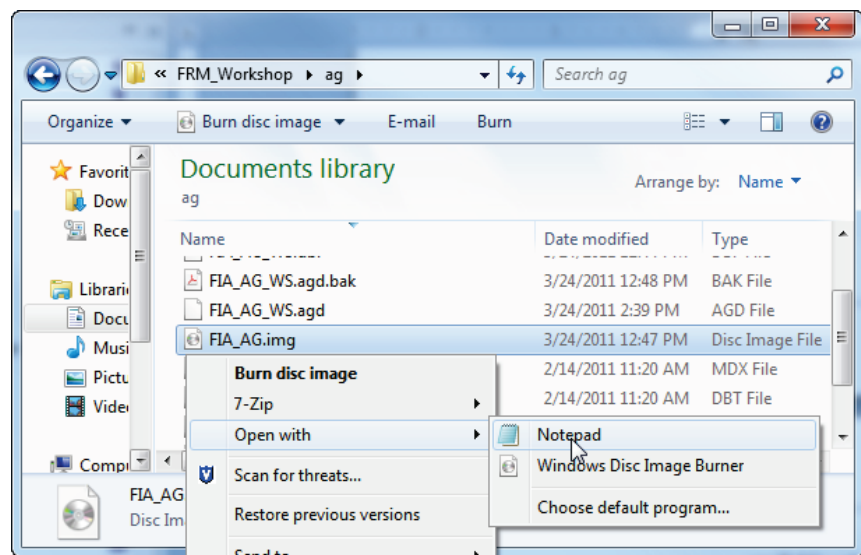
Figure 7.5 Ag folder (FIA Study Folder).

2. From the **Choose Program** dialog scroll down until you find **Notepad**, select it and press **OK** to open the .img file (Figure 7.7).
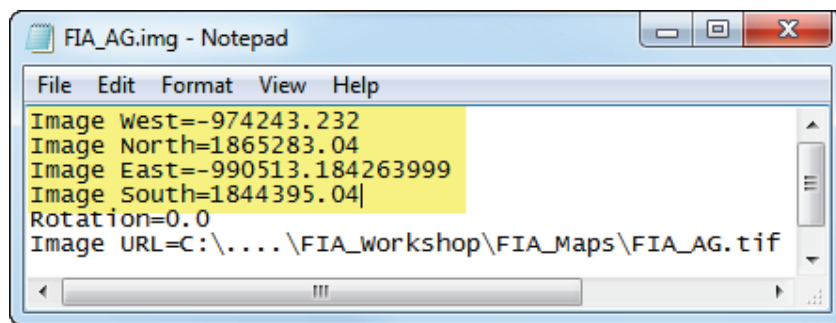
Figure 7.6 Agricultural spatial reference in .img file.

3. The .img file contains spatial reference information for the agricultural grid data in HEC-FIA. In this case, the extents were not converted to feet from meters. Convert the Image West, North, East, and South values to English units and save the file (Figure 7.8).
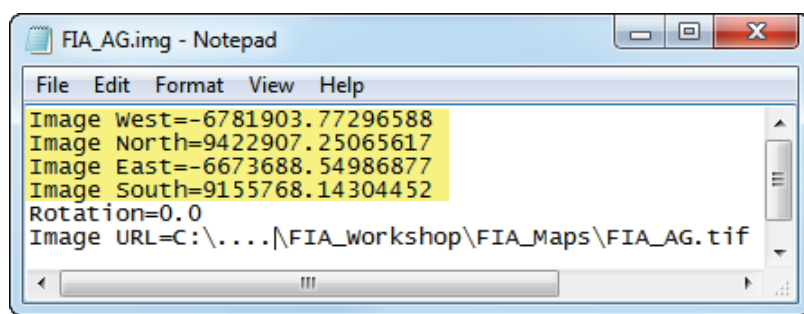
Figure 7.7 Agricultural spatial reference converted to feet.

4. From the **File** menu, click **Save Study**, or from the toolbar click 🖫.
5. From the **File** menu, click **Exit** to close FIA.
6. Reopen HEC-FIA and from the **File** menu, click **Recent Studies** to reopen the study. Your agricultural data should now display properly (Figure 7.9).
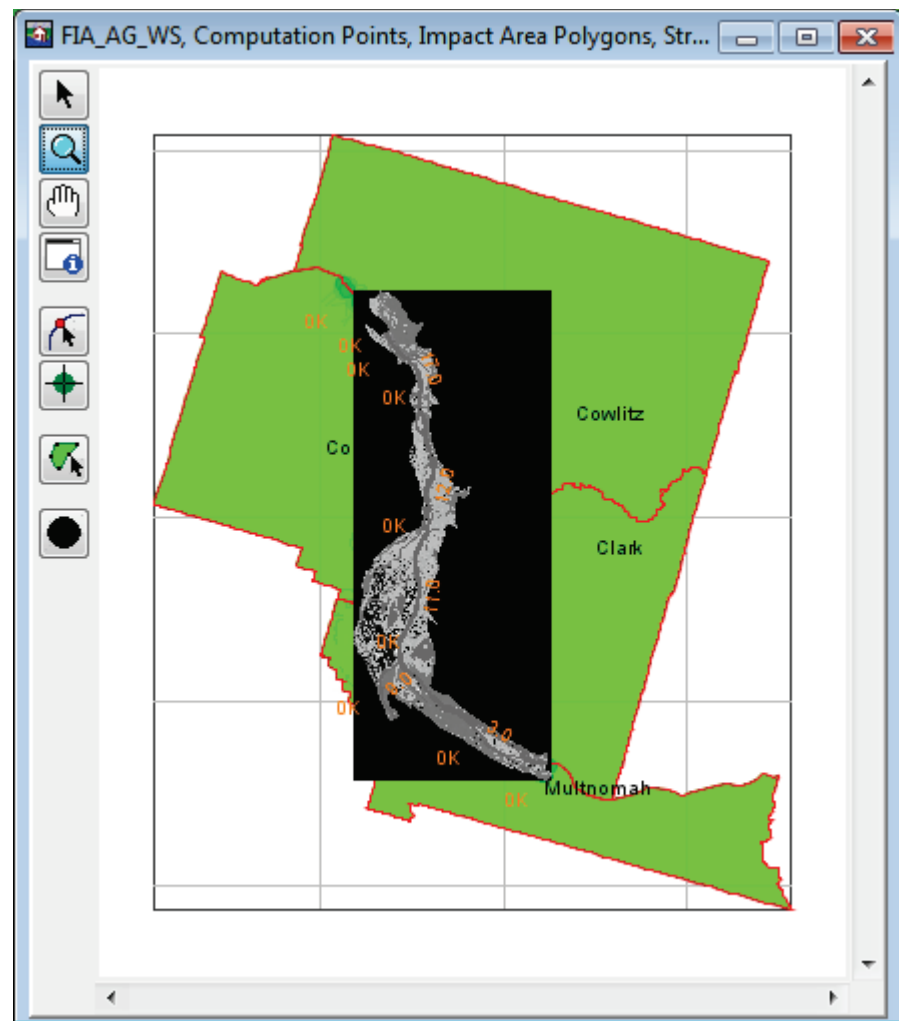
Figure 7.8 After extents are edited, your agricultural data should properly display in the Map Window.

7.1.2   Import Agricultural Data from Shapefile

Users may import agricultural data from a shapefile that represents crop coverage for their study. Before the importation steps below, upload the agricultural shapefile to your study (see Chapter 3 for adding map layers to a study). To import the agricultural data into FIA, follow the steps below.

1.   From the **Study Pane**, right click on the **Agricultural Grids** folder. From the shortcut menu (Figure 7.1), choose **Generate From→ Shapefile**.
2.   The **Generate From Shapefile** dialog box will appear (Figure 7.9).
3.   In the **Name** box, enter a name for the imported agricultural data. In the **Directory** box browse ⬚ to the location where you would like to store the imported data.
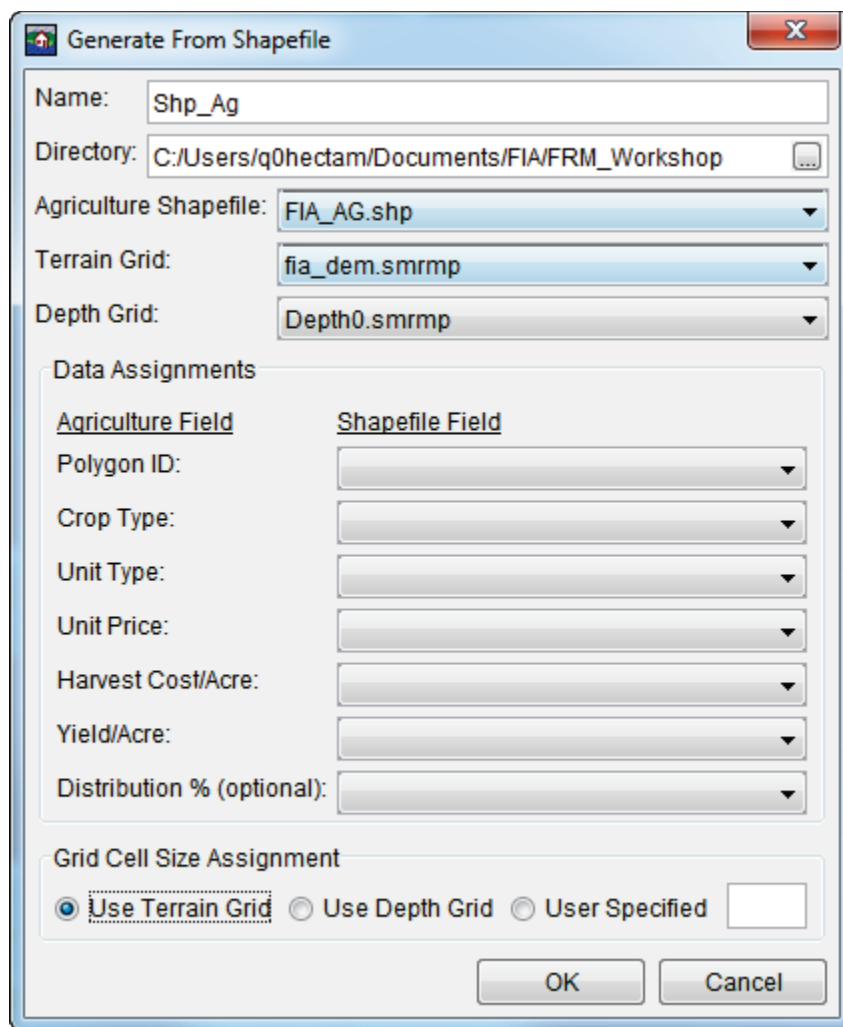
Figure 7.9 Generate From Shapefile dialog box.

4. From the **Agricultural Shapefile** list, select the shapefile uploaded to your study that contains agricultural data. Select the **Terrain Grid** and maximum inundation depth grid (**Depth Grid**) from the dropdown lists.

5. Under the **Data Assignments** section is a list of crop data in one column and dropdown lists representing the data fields within the agricultural shapefile in the other column. From the column on the right, associate the appropriate **Shapefile Field** for each **Agriculture Field** element on the left. All fields except for **Distribution %** are required. For an explanation of the **Agriculture Field** terms, see Section 7.4.

6. At the bottom of the dialog, select the desired grid cell size for the agricultural data. Users can **Use Terrain Grid** for the grid cell size of the terrain grid, **Use Depth Grid** for the grid cell size of the depth grid, or choose their own **User Specified** grid cell size by entering a value in the box.

7. Click **OK**. The dialog will close and the name of your new agricultural grid will appear under your new agricultural grid will

appear under the **Agricultural Grids** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved (Figure 7.3).

8.   To save your study and your new agricultural grid, from the toolbar click ▣ .

7.1.3   Import Agricultural Data from HAZUS

Users can also import agricultural data using the HAZUS agricultural database they prepared for their study (i.e. fiAG.mdb; see Appendix B, Section B.6) and a shapefile that represents crop coverage for their study.

1.   From the **Study Pane**, right click on the **Agricultural Grids** folder. From the shortcut menu (Figure 7.1), choose **Generate From➞ Hazus database**.

2.   The **Generate Crop Data From HAZUS Database** dialog box will appear (Figure 7.10).
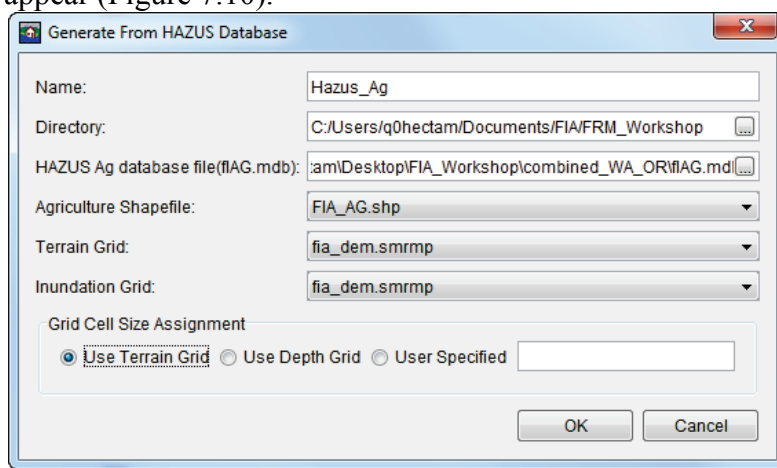
Figure 7.10 Generate From HAZUS Database dialog box.

3.   In the **Name** box, enter a name for the imported agricultural data. In the **Directory** box browse 🔲 to the location where you would like to store the imported data. In the **HAZUS Ag database file(fiAG.mdb)** field, click browse 🔲 to browse to the location of the HAZUS agricultural database file (fiAG.mdb) for your study. An **Open** browser will open (Figure 7.11). From the **Files of type** list at the bottom of the browser, change the file type from **HAZUS Ag database file (fiAg.mdb)** to **All Files**. Now you will be able to see all Microsoft Access database files. Locate and select the HAZUS agricultural data, by convention named *fiAG.mdb*. At the bottom of the browser, click **Open**. The browser will close and the name of your HAZUS agricultural database will appear in the field.

4.   From the **Agricultural Shapefile** list, select the shapefile uploaded to your study that contains agricultural data. Select the **Terrain Grid** and maximum inundation depth grid (**Inundation Grid**) from the dropdown lists.
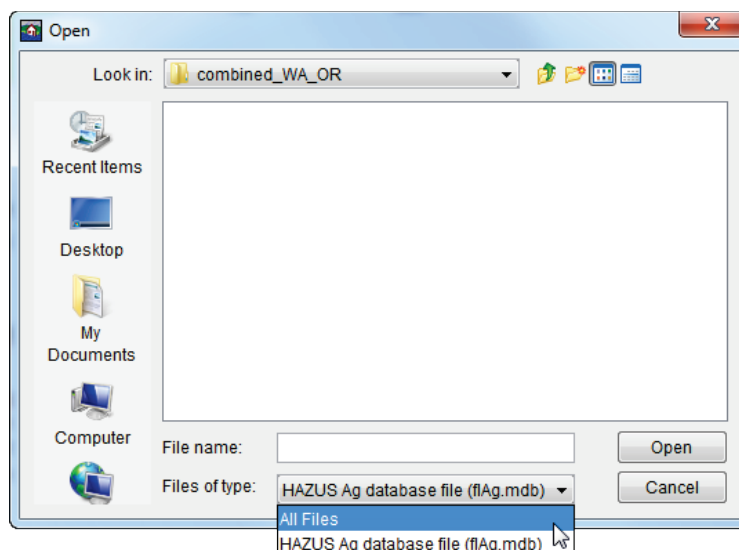
Figure 7.11 Open browser.

5. At the bottom of the dialog, select the desired grid cell size for the agricultural data. Users can **Use Terrain Grid** for the grid cell size of the terrain grid, **Use Depth Grid** for the grid cell size of the depth grid, or choose their own **User Specified** grid cell size by entering a value in the box.

6. Click **OK**. The dialog will close and the name of your new agricultural grid will appear under your new agricultural grid will appear under the **Agricultural Grids** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved (Figure 7.3).

7. To save your study and your new agricultural grid, from the toolbar click 🔲 .

7.2   **Remove or Delete Agricultural Data**

To remove agricultural data from your study:

1. Starting in the **Study Pane**, under the **Agricultural Grids** folder locate the name of the agricultural grid you wish to remove.

2. From the shortcut menu, select **Remove From Study** (Figure 7.12).
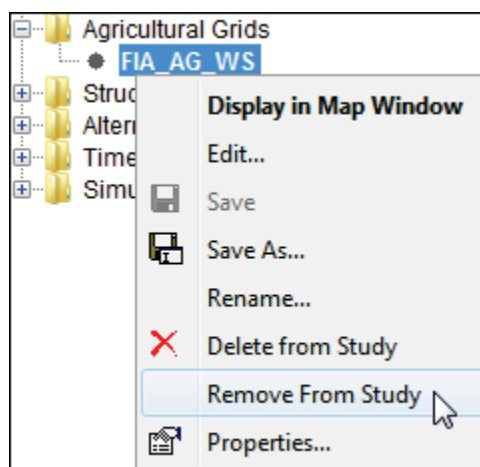
Figure 7.12 Agricultural grid shortcut menu (Study Pane).

3. The **Confirm Removal** dialog box will appear, prompting the user to confirm removal of the grid. To remove, select **Yes**.
4. Your agricultural grid will now be removed from the study.

Selecting the **Remove From Study** option will only remove the selected agricultural data from a given study, not remove that data from your hard drive. This is useful if you would like to retain the .img file, for example, of a NASS CDL grid with edited extents, but not use that data in your study.

If instead you wish to delete the agricultural grid from your hard drive:

1. Starting in the **Study Pane**, under the **Agricultural Grids** folder locate the name of the agricultural grid you wish to delete.
2. From the shortcut menu (Figure 7.12), select **Delete from Study**.
3. The **Confirm Manager Delete** dialog box will appear, prompting the user to confirm removal of the grid. To delete the grid and its associated files, select **Yes**.
4. The agricultural grid will now be deleted from your hard drive.

7.3   Rename Agricultural Data

From the **Agricultural Grid shortcut** shown in Figure 7.10, users can also rename an agricultural grid.

To rename an agricultural grid:

1. Starting in the **Study Pane**, under the **Agricultural Grids** folder locate the name of the grid you wish to rename.
2. From the shortcut menu (Figure 7.12), select **Rename**.
3. The **Rename Agricultural Grids** dialog box will appear. Enter a new name for your grid and a short description.
4. Another dialog box will appear, prompting the user to confirm renaming of the file (Figure 7.13).
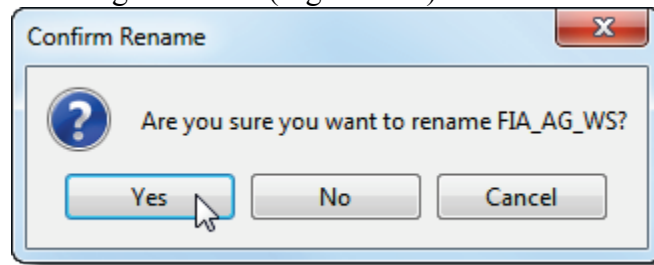
Figure 7.13 Confirm Rename dialog box.

5. Select **Yes** to rename. Select **No** to cancel renaming and return to your study. Select **Cancel** to return to the **Rename Agricultural Grid** dialog and make a change in name or description.

6. Your newly-renamed grid will appear under the **Agricultural Grids** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.

7. To save your study and your newly renamed grid, from the toolbar click ⊞.

7.4 Edit Agricultural Data

Once an agricultural grid has been loaded, users must define crop loss function data so that FIA can compute damages. This is accomplished by defining a **crop loss function** and entering **seasonal data** used to estimate the damage of the crop at a given time of year.

Definition of data entry terms are as follows:

- **Yield Per Acre** - this is the average number of units produced per acre of land on which the crop is planted when the crop is harvested.
- **Unit** – is used to measure the amount of harvested crop. For example, corn may be measured in bushels.
- **Unit Price ($)** – value in dollars of a single unit of harvested crop.
- **Harvest Cost ($)** - the amount in dollars it costs to harvest one acre of the crop.
- **Crop Loss Function** defines the crop damage potential for a specific crop as it varies throughout the year from soil preparation to completion of harvest

7.4.1 Define Crop Loss Function

A **crop loss function** defines the crop damage potential for a specific crop as it varies throughout the year from soil preparation to completion of harvest. FIA includes a set of predefined crop loss functions for a variety of crops which are also user editable; these defaults were obtained from HAZUS and are general crop loss functions. The user should not use these defaults unless there is no way for obtaining region specific data, in which case justification for using the defaults should be included in any report containing results.

To define a **crop loss function** in HEC-FIA:

1. Starting in the **Study Pane**, under the **Agricultural Grids** folder locate the name of the grid for which you wish to define crop loss data and right-click. From the shortcut menu (Figure 7.12), select **Edit**.

2. The **Define Crop Loss Function** dialog box will appear (Figure 7.14).

Figure 7.14 Define Crop Loss Function dialog box.

3. Enter the Yield per Acre, Unit, Unit Price, and Harvest Cost for each crop in the study area in the provided table. Depending on the source of your agricultural grid, some of these fields may be entered for you. Most of this information can be found on the NASS website in the Statistics by State section: http://www.nass.usda.gov/Statistics_by_State/

4. From the **Crop Loss Function** list, choose the appropriate function for each crop. In the example in Figure 7.15, the Alfalfa is being selected as the crop loss function for Alfalfa.

Figure 7.15 Select Crop Loss Function.

5. FIA contains a variety of preset crop loss functions, and all are user-editable for each crop via the **Edit Function** button.

To modify a crop loss function:

    a. You are still in the **Define Crop Loss Function** dialog box (see step 1 of this section). Locate the crop for which you wish to edit a **Crop Loss Function**.

    b. Select a function from the dropdown list (Figure 7.15).

    c. Press the **Edit Function** button (Figure 7.16).

    d. The **Crop Loss Function** dialog box (Figure 7.17) will open, which includes the name of the function you are editing in the title. The table shows percentage of crop loss for floods of duration 0, 3, 7, and 14 days on each calendar day of the year. In the table, enter the numbers that will create a damage function for your crop. For help determining the production functions and duration damage relationships contact your local land grant university or agriculture extension agency. http://www.csrees.usda.gov/Extension/

Figure 7.16 Edit Crop Loss Function (Define Crop Loss Function dialog box).
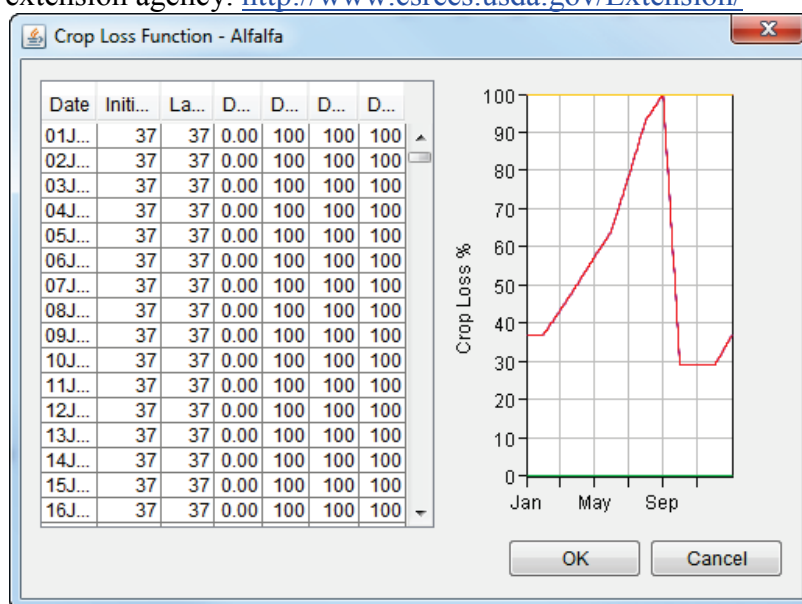
Figure 7.17 Crop Loss Function dialog box.

    e. As values are entered, the thumbnail plot to the right of the table will continually update.

    f. When you are finished editing your crop loss function, click **OK.** You will now be back in the **Define Crop Loss Function** dialog box.

6. Once a crop loss function is defined, users must enter **seasonal data**; see the following section 7.5.2.

7.4.2   Enter Seasonal Data

Seasonal data is required to estimate the amount of damage to a crop at a given time of year. If a flood occurs well before a farmer plants their crops, then there may be little to no damage done. If that same farm gets flooded prior to harvest and after planting, the damage could be substantial.

Seasonal crop data includes: first day of planting, last day of planting which will produce a full yield, last day of planting which will produce a partial yield, and the last day of harvest.

- **First Plant Date** – first date that the crop can be planted.
- **Full Yield Date** – latest date that the crop can be planted and still reach full maturity.
- **Last Planting Date** – latest date that the crop can be planted with a reduction in yield.
- **End of Harvest Date** – last date that the crop is harvested, after which little damage can be incurred.
- **Dryout Period (days)** – number of days after a flood has receded before the soil is sufficiently dry so replanting can begin.

To enter Seasonal Data for a given crop:

1. You are still in the **Define Crop Loss Function** dialog (see section 7.4.1, page 7-10).
2. Locate the name of the crop for which you want to enter **Seasonal Data**.
3. Locate the **Seasonal Data** column and select the **Enter** button (Figure 7.18).
4. The **Seasonal Data** dialog box will open (Figure 7.19). Click the ⬚ button to enter the First Plant Date, Full Yield Date, Last Planting Date, End of Harvest Date, a Dryout period, and the percentage loss if planting occurs on the Last Plant Date. Dates must be entered in *ddmmm* month format (e.g. *08Feb*)
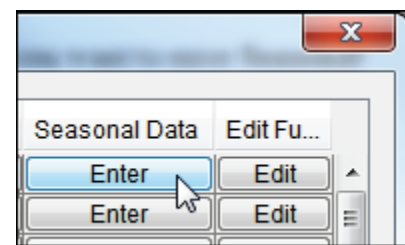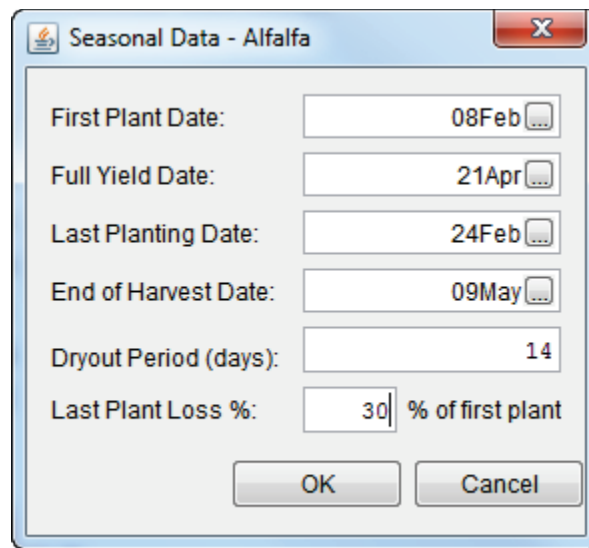
Figure 7.18 Enter Seasonal Data

3. Click **OK**.

Figure 7.19 Seasonal Data dialog box.

4. You will now be back in the **Define Crop Loss Function** dialog.
5. A **crop loss function** and **seasonal data** must be entered for HEC-FIA to compute damages for each crop. Repeat steps 1-6 in section 7.4.1 and 1-6 in section 7.4.2 until each crop in your agricultural grid is defined.
6. If you are finished entering **crop loss functions** and **seasonal data** for your study, in the **Define Crop Loss Function** dialog box click **OK**. The dialog will close.
7. To save your study and your newly defined agricultural grid, from the toolbar click ⊞ .

Chapter 8

Structure Inventories

8.1   Create a Structure Inventory

A structure inventory is required to estimate direct economic damages for a flood event. To estimate life loss for an event, a structure inventory must be supplied with population. HEC-FIA can create structure inventories in the form of HAZUS databases, point shapefiles, and parcel shapefiles.

8.1.1   Generate a Structure Inventory from HAZUS Data

Typically, structure inventories with population will be generated using the databases supplied with the HAZUS data. An important note regarding HAZUS data is that the data is generated by FEMA using statistical processes based on regional data. This leads to potential issues in the database that might yield results that are unreasonable. The user should insure that their data is actually representative of the study area.

To generate a structure inventory using HAZUS data:

1. From the **Study Tree**, in the **Structure Inventory** folder, right-click on **Global Inventory**. From the shortcut menu (Figure 8.1), point to **Generate From** and click **Hazus database**.
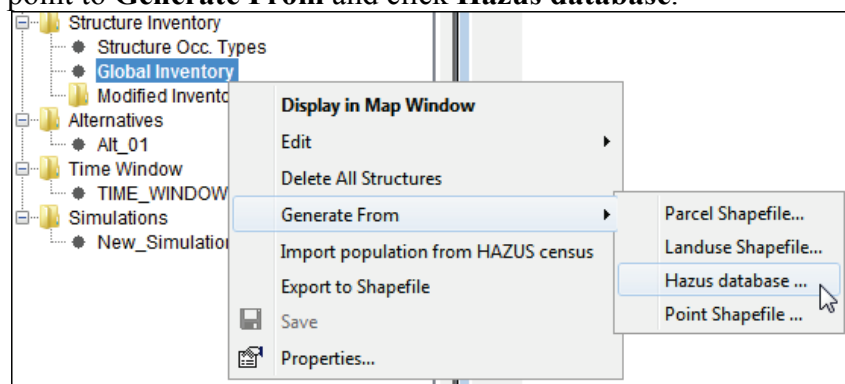
Figure 8.1 Global Inventory shortcut menu (Study Pane).

2. The **Generate Structure Inventory from Hazus Database** dialog box will open (Figure 8.2).
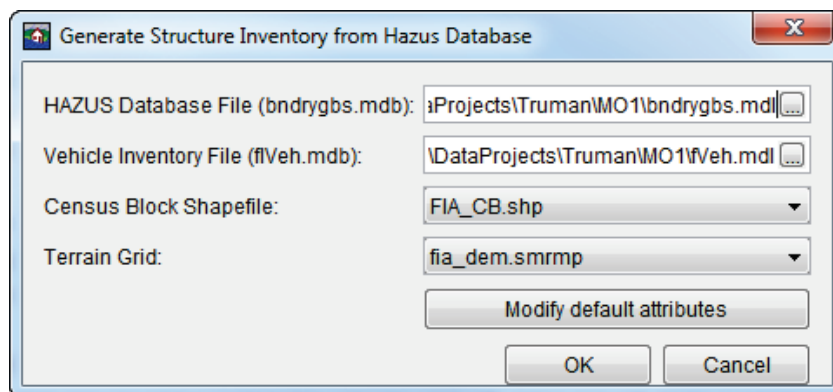
Figure 8.2 Generate Structure Inventory from Hazus Database dialog box.

3. In the **HAZUS Database File (bndrygbs.mdb)** box, browse to the location of the required file by clicking ⬚. An **Open** browser will be invoked. Browse to the file that contains the HAZUS database file (e.g., *E:\DataProjects\Truman\MO1*), select the filename (i.e. *bndrygbs.mdb*), and click **Open**. The path and filename of the HAZUS database file will appear. All HAZUS databases have the same file structure and naming convention so bndrygbs.mdb should be a consistently chosen file name for the HAZUS database file.

4. In the **Vehicle Inventory File (flVeh.mdb)** box, browse to the location of the file by clicking ⬚. An **Open Browser** dialog will be invoked. Browse to the location of HAZUS database file (e.g., *E:\DataProjects\Truman\MO*1), select the filename (e.g. *flVeh.mdb*), and click **Open**. The path and filename of the vehicle inventory file will appear. As with the bndrygbs.mdb file, the naming convention of the HAZUS vehicle inventory data flVeh.mdb will be consistent with all HAZUS databases.

5. From the **Census Block Shapefile** list, select the shapefile that contains the census block data for the study area (e.g., *FIA_CB.shp*). Refer to Chapter 3 for instructions on how to add shapefiles to an FIA study.

6. From the **Terrain Grid** list; select the terrain grid for the study area (e.g., *fia_dem.smrmp*). Refer to Chapter 6 for instructions on how to add a terrain grid to an FIA study.

7. At this point, HEC-FIA allows users to globally edit certain attributes of structure inventory data before importation by using the **Modify Default Attributes** function.

    a. The **Default Attributes** dialog box will appear (Figure 8.3).
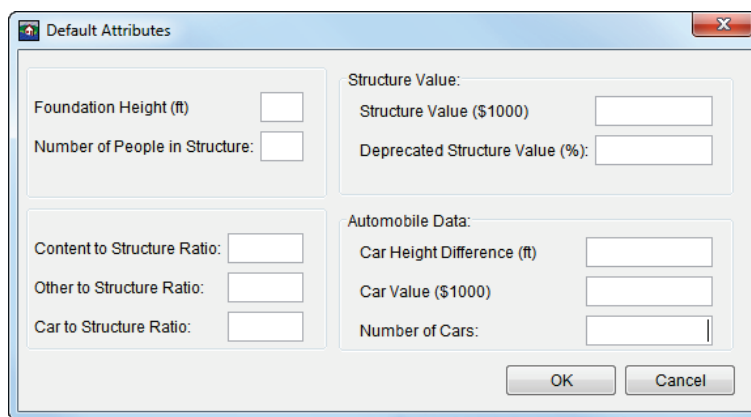
Figure 8.3 Default Attributes dialog box.

    b. Data entry fields include **Foundation Height**, **Number of People in Structure**, **Content to Structure Ration**, **Other to Structure Ratio**, **Car to Structure Ratio**, **Structure Value**, **Depreciated Structure Value**, **Car Height Difference**, **Car Value**, and **Number of Cars**.

    c. The data entered here will apply to every structure in an inventory.

    d. After making the desired changes, select **OK**. The **Default Attributes** dialog box will close, leaving the Generate Structure Inventory from Hazus Database dialog still open.

7.   Click **OK**. A progress dialog box will appear (Figure 8.4). This dialog box shows the status of the structure generation and when the structure inventory has been generated. The operation may be aborted by pressing **Cancel**. When the import process is complete, the dialog will close and all of the structures associated with the study area will appear in the **Map Window**.
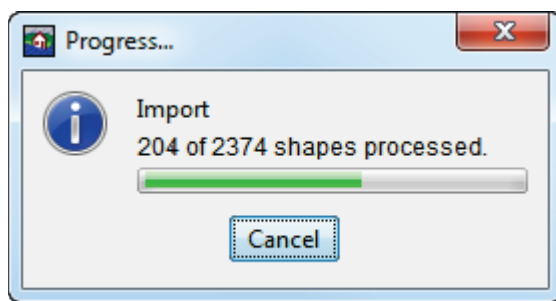
Figure 8.4 Import progress dialog box.

8.   Another progress dialog box will appear, displaying the status of the population distribution. When the population distribution has been generated, the progress dialog box will close.

8.1.2   Generate Structure Inventory from Point Shapefile

To generate a structure inventory from a point shapefile:

1. From the **Study Tree**, in the **Structure Inventory** folder, right-click on **Global Inventory**. From the shortcut menu, point to **Generate From** and select **Point Shapefile** (see Figure 8.1).
2. The **Generate Structure Inventory from Point Shapefile** dialog box will open (Figure 8.5).

Figure 8.5 Generate Structure Inventory from Point Shapefile dialog box.

3. In the **Shapefile** field, select the shapefile you would like to associate with structure data (e.g. *R1_SI.shp*) from the dropdown list. In the **Terrain Grid** box, select the SMRMP file of your study's terrain grid (e.g. *r01_5m.smrmp*).
4. The four fields under **Data Assignments** are required for calculation of structure damages. Select the appropriate **Shapefile Field** name associated with each **Inventory Field**.
5. The additional fields under **Optional Data** are for data that is not required to calculate structure damages but may still be important to the study (e.g., life loss, content value, foundation height). Within

the dropdown lists in the **Inventory Field** column, you will find a list of structure attributes akin to those found in the **Structure Editor**. To assign shapefile attributes to the FIA inventory, choose an inventory field, and under the **Shapefile Field** column select the **Shapefile Field** that contains the data associated with the structure attribute on the left.

6. Once you have filled out the above fields, click **OK**. A dialog box will appear displaying the progress of shapefile import (just as in Figure 8.4). This dialog box shows the status of the structure generation. When the import process is complete, the dialog will close and all of the structures associated with the study area will appear in the **Map Window**.

8.1.3   Generate Structure Inventory from Parcel Shapefile

To generate a structure inventory from a parcel shapefile:

1. From the **Study Tree**, in the **Structure Inventory** folder, right-click on **Global Inventory**. From the shortcut menu, point to **Generate From** and select **Parcel Shapefile** (see Figure 8.1, page 8-1).
2. The **Generate Structure Inventory from Parcel Shapefile** dialog box will open (Figure 8.5). Aside from the title, this dialog is exactly the same as that for point shapefiles.
3. Complete Steps 3 through 6 from the previous section 8.1.2.

8.1.4   Manual Entry by Structure Tool

Users may also manually create structures using the **Structure Tool**. To access the **Structure Tool**, double-click on **Global Inventory** in the **Study Pane** to make sure it is displayed in the **Map Window**. In the **Map Window**, engage the **Structure Tool** and while holding the Ctrl button, click at the location where you would like to create a structure.

8.1.5   Index Curves

When using Time-Series HEC-DSS Hydrographs to compute damages, users may create a structure inventory under the **Damage Functions** tab of the **Impact Area Editor** using aggregated index curves. This functionality is covered in Chapter 9, section 9.7.6.

8.2   Edit a Structure Inventory

For precision in calculation of structural damages, FIA allows users to enter a wide variety of data for individual structures, groups of structures, or globally, as in the **Modify default attributes** function (see Figure 5.4). The user can choose the level of detail and effort they wish to use to

calculate structure damages by editing their structure inventory. This section will explore the tools for editing structure inventories in HEC-FIA.

The **Structure Editor** is the primary tool for entering detailed information into a structure inventory. Within it are tabs for modifying the **Location**, **Classification**, **Values**, **Elevations**, **Address**, and **Attributes** of a given structure.

The following sections will cover navigation of the Structure Editor as well as the functions located within each tab.

8.2.1   Open the Structure Editor

There are several ways to access the **Structure Editor:**
1. In the map window, select the **Structure Tool** and right-click on the structure you want to edit. From the shortcut menu, select **Edit►Structure Form** (Figure 8.6).
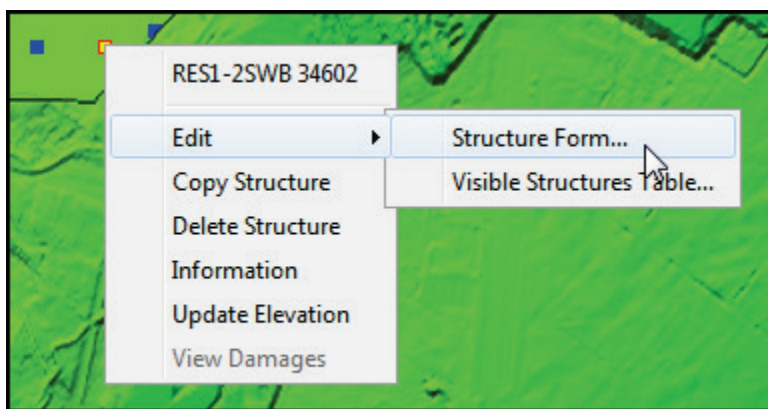
Figure 8.6 Structure shortcut menu (Map Window).

2. Starting in the Study Pane, right-click on **Global Inventory** and select **Edit⟶Form Editor** from the shortcut menu (Figure 8.7).
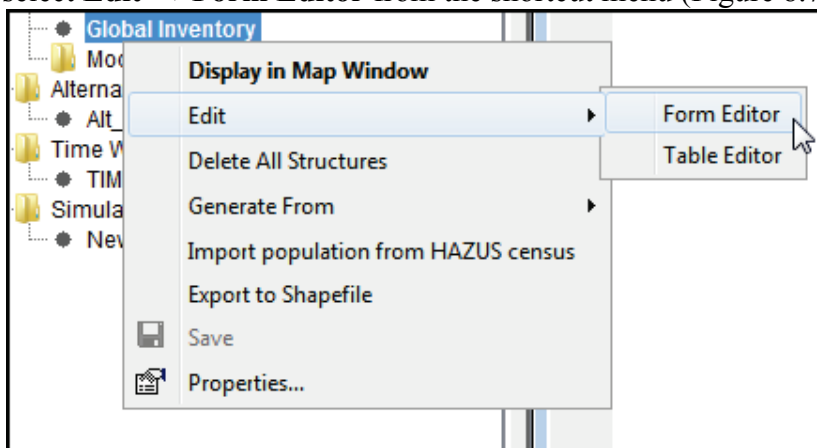
Figure 8.7 Global Inventory shortcut menu (Study Pane).

3. Alternatively, users may access the Structure Editor under the **Data** menu by selecting **Structure Inventory→Form Editor.**

Completing any of the preceding steps will open the **Structure Editor** (Figure 8.8).
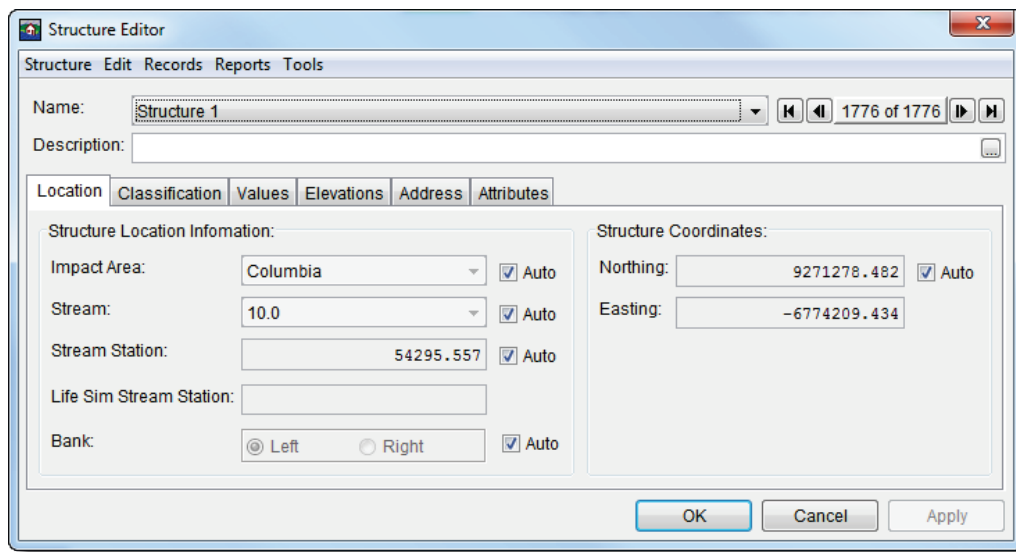
Figure 8.8 Structure Editor.

8.2.2   Navigating the Structure Editor

This section covers navigation of the basic elements of the **Structure Editor**. The **Menu Bar** (Figure 8.9) at the top of the editor allows users to perform basic functions like creating, renaming, or deleting individual structures. Other features, like viewing structure information in report form, are also available from the **Menu Bar.**
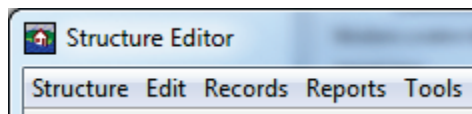
Figure 8.9 Structure Editor—Menu Bar.

The main content of the **Structure Editor** is found in its six tabs, **Location**, **Classification**, **Values**, **Elevations**, **Address** and **Attributes**. These Tabs are used to enter detailed information about each structure. The following sections will cover how to create, rename, and delete individual structures as well as the many functions of the various tabs.

8.2.3   Create a Structure

To create a structure using the **Structure Editor**, click on the **Structure** menu under the **Menu Bar**. From the menu, select **New**. In the **Create New Structure** dialog box, enter a name (required) and a short description.

8.2.4   Delete a Structure

To delete a structure using the **Structure Editor**:

1. Using the **Name** dropdown list, scroll to the name of the structure you would like to delete (Figure 8.10). A box to the right of the dropdown list will display the numerical position of that structure within your entire inventory (Figure 8.8).
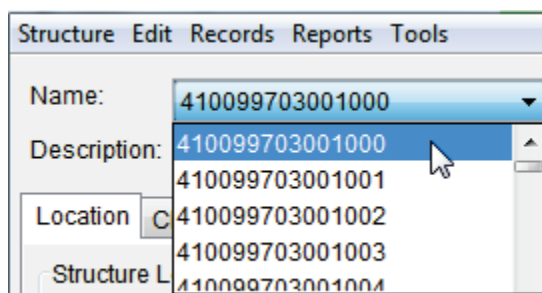
Figure 8.10 Name list (Structure Editor).

2. From the **Structure Menu**, select **Delete**.
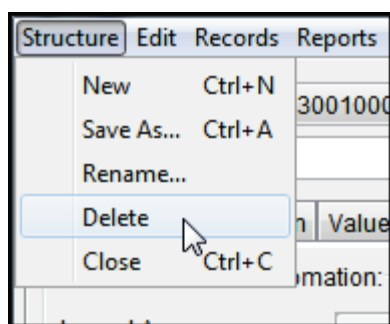
Figure 8.11 Structure Editor—Structure Menu.

3. A dialog box will appear, asking you to confirm deletion of the structure. To delete, select **Yes**.

Alternatively, users may delete individual structures in the **Map Window** using the **Structure Tool** 🏠.

1. Starting in the **Map Window**, select the **Structure Tool**.
2. Right-click on the structure you wish to delete.
3. From the shortcut menu, select **Delete Structure**.
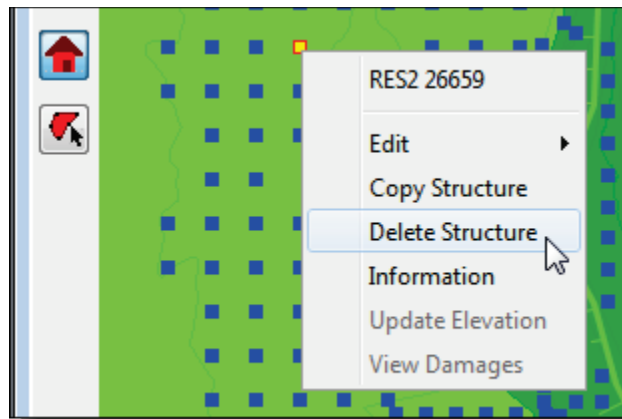
Figure 8.12 Structure shortcut menu (Map Window).

4. A dialog box will prompt you to confirm deletion. To delete, select **Yes**.

8.2.5   Rename a Structure

To rename a structure in the **Structure Editor**, click on the **Structure Menu** (Figure 8.11) and select **Rename**. In the **Rename Structure** dialog, change the structure name and description, if desired.

8.3   Location Tab

The **Location** tab allows users to edit information about the location of a given structure (Figure 8.13). By default, HEC-FIA calculates this information automatically. The information may be entered manually by clearing the ☑ Auto box and entering a new value.
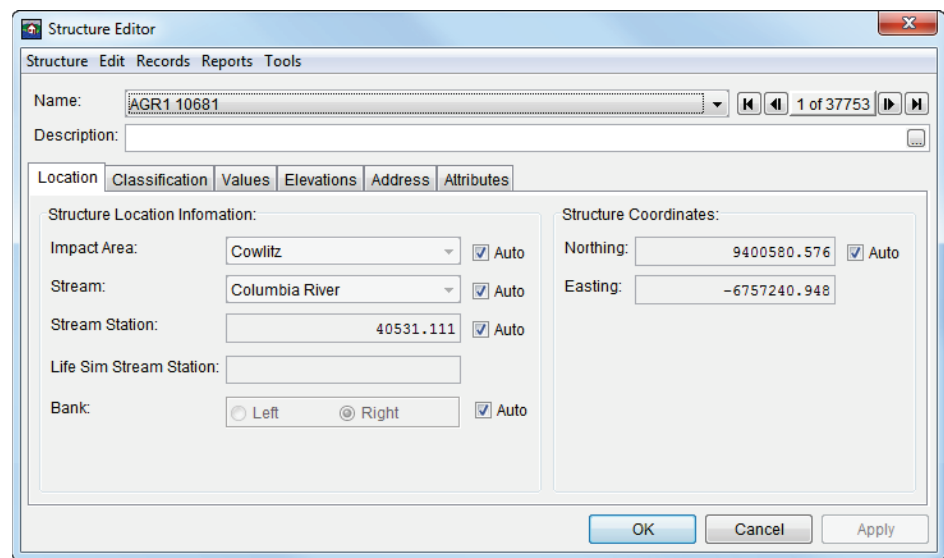
Figure 8.13 Structure Editor—Location tab.

8.4   Classification Tab

The **Classification** tab allows users to view the damage categories and functions that HEC-FIA will use to compute damages for a given structure (Figure 8.14**).** Within the **Classification** tab**,** users can select a **Damage Category** and a **Structure Occupancy Type**. Usually FIA will assign damage categories and structure occupancy types to a structure inventory during the import process, but these assignments are user-editable under the **Classification Tab**.

Figure 8.14   Structure Editor—Classification tab.

8.4.1   Assign Damage Categories

A **Damage Category** is a consolidation of structure types into specific categories with similar characteristics for analysis and reporting. HEC-FIA contains pre-defined damage categories (Commercial, Industrial, Public, and Residential) and allows users to create their own in the **Global Data Editor** (see Chapter 4). To assign a damage category, browse through the dropdown list and choose the category you wish to associate with the selected structure (Figure 8.15).
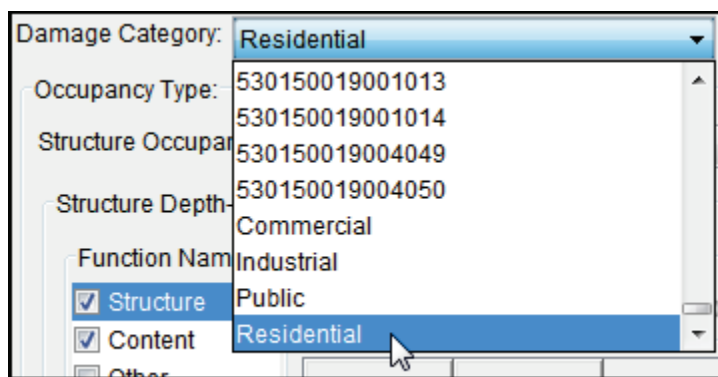
Figure 8.15 Assign a damage category (Structure Editor—Classification tab).

8.4.2   Assign Structure Occupancy Types

The **Structure Occupancy Type** defines the damage function that HEC-FIA will use to compute damages. In the **Classification** tab, users can assign a **Structure Occupancy Type** to a structure. To edit a **Structure Occupancy Type,** refer to the following section 8.10.

To assign a **Structure Occupancy Type**, browse through the dropdown list and select the appropriate item (Figure 8.16). The values of the **Structure Depth-Damage Function** table at the bottom-center of the **Structure Editor** will change accordingly.
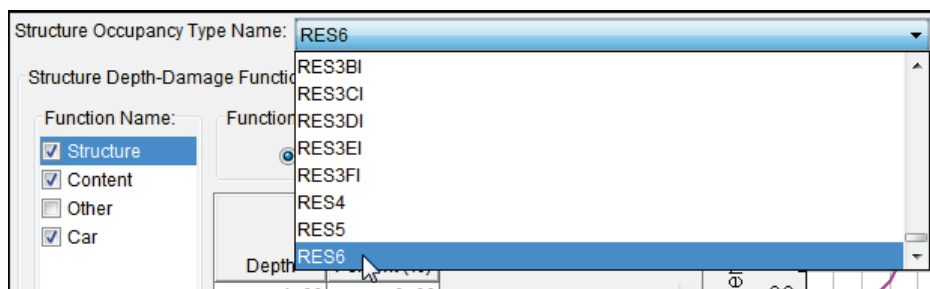
Figure 8.16 Assign a Structure Occupancy Type (Structure Editor—Classification tab).

8.4.3   Structure Depth-Damage Functions

The remaining portion of the **Classification** tab allows users to view the **Depth-Damage Function** associated with a particular structure. The **Structure Depth-Damage Function** is shown in chart form as inundation depth minus foundation height against percentage of damage. This function is plotted to the right of the table in a thumbnail plot (Figure 8.17). To view a full-size plot, press the [Plot...] button.
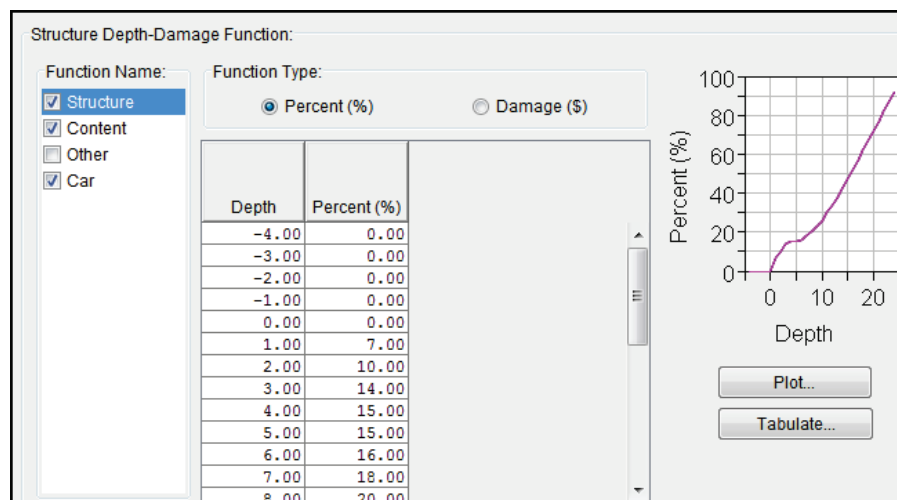
Figure 8.17 The Structure Depth-Damage Function worksheet (Structure Editor—Classification tab).

Up to four different damage functions can be associated with a **Structure Occupancy Type** to allow for damage to the structure itself, the contents of the structure, a vehicle associated with it, and a user-defined damage function. To view a given function, select its name under **Function Name**.

Note that users may only *view* the **Structure Depth-Damage Function** in the **Structure Editor**; to *edit* the function, see **Section 8.10**.

8.5   Values Tab

Under the **Values** tab, users can assign numerical values to a variety of fields (Figure 8.18). The monetary value of the structure, its contents, vehicles, and an "Other" user-defined category can be entered in thousands of dollars. Population of the structure at different times of day and by age demographic, the year the building was built (**Year in Service**) and number of structures within it may also be entered. These data will be to estimate the economic damages and life loss impact of a flood event. **Year in Service** is for information only and will not affect compute. Users may view and search the data entered under the **Values Tab** by accessing the report under **Reports━▶Structure – Values and Information Report**.

Figure 8.18 Structure Editor—Values tab.

8.6   Elevations Tab

Under the **Elevations** tab, users enter information about the elevation of a structure. A chart shows users what elevation each field modifies (Figure 8.19).

Figure 8.19 Structure Editor—Elevations tab.

Definition of data entry terms as follows:
- **First Floor Elevation** is the elevation of the first level of the structure, defined as Ground Elevation + Foundation Height.
- **Ground Elevation** is the elevation of the terrain at the location of the structure.
- **Foundation Height** is the height of the structure's concrete foundation, defined as the difference between First Floor Elevation and Ground Elevation.

- **Beginning Damage Depth** is the elevation at which flood waters begin entering the structure.
- **Depth Difference for Cars** is the difference between the **Beginning Damage Depth** for structures and the elevation at which damage for vehicles begins. Typically this will be the same as foundation height.

By selecting ⊙ First Floor Elevation (ft), users may manually enter the elevation of the first floor of the house. Selecting ⊙ Ground Elevation will allow users to enter the structure's ground elevation and foundation height to calculate the elevation of the first floor. FIA will use the terrain grid to calculate ground elevation for each structure. To manually enter a ground elevation value, clear the ☑ Auto box and enter a new value. Users may enter the foundation height just beneath it.

Users can view elevations information about their structure inventory with the **Elevations Information Report. View** the **Elevations Information Report** by selecting **Reports⟶Structures – Elevations Information Report.**

8.7   Address Tab

Under the **Address** tab, users enter address information of structures (Figure 8.20). Address information is for information only and does not affect compute. In this version of FIA, the image upload function is not functional. Access the **Address Information Report** by selecting **Reports⟶Structures – Address Information Report** from the **Menu Bar** of the **Structure Editor**.

Figure 8.20 Structure Editor—Address tab.

8.8   Attributes Tab

The **Attributes** tab is organized into four sections of data input: **Basic Parameters**, **Tax Assessor Data**, **NAICS Classification**, and **Construction Type**. These allow users to enter detailed information about various aspects of the structure (Figure 8.21). These data are unnecessary for a typical study, but FIA allows users to enter information to this level of detail if desired.

Figure 8.21 Structure Editor—Attributes tab.

Under **Basic Parameters**, users enter detailed information about the structure. Data entry fields include: **Total Floor Area (sq ft)**, **Number of Stories**, **Effective Age**, **Condition** (select from the dropdown list), **IRIS Land Use**, and **Basement Floor Area (sq ft)**.

Under **Tax Assessor Data**, enter tax assessment information about the building. This information is usually obtained on a given county assessor's website. Search online for the county's assessment office and look for an option for parcel number search. You will usually be able to search by the address of a structure, and the query will return results for the structure's **Parcel Number** (an identification number assigned to properties), **Assessed Value**, and the year it was assessed (**Year Assessed**).

Under **NAICS Classification**, users select a **NAICS Code** to assign to their structure. The North American Industry Classification System

(NAICS) is the United States statistical agencies standard method for collecting statistical data on the business economy of the United States. Thus a NAICS code is used only for commercial and industrial buildings. To assign a NAICS code to a structure:

1. Starting in the **Attributes** tab of the **Structure Editor**, click **Select Code**.
2. The **Select NAICS Code** dialog box will appear (Figure 8.22).
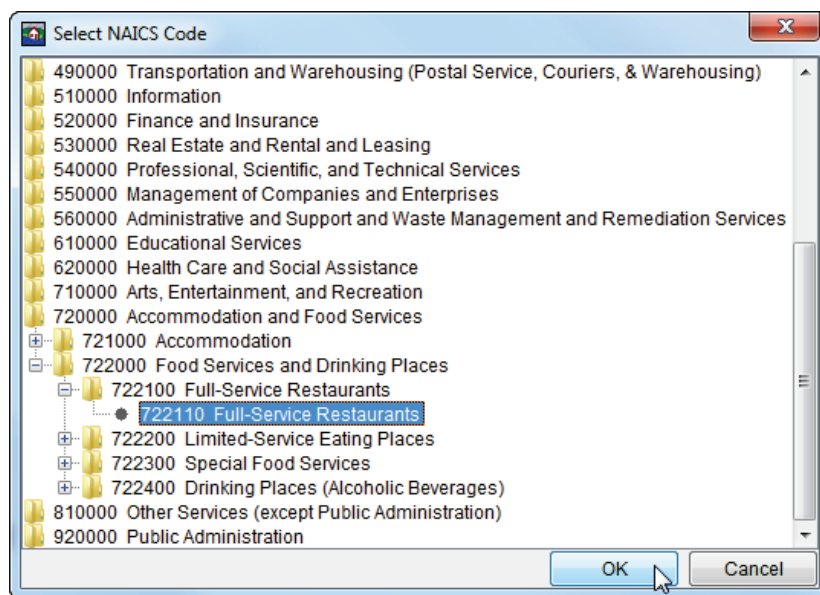
Figure 8.22 Select NAICS Code dialog box (Structure Editor—Attributes tab).

3. Browse to the category that describes the selected structure and double click. For each main category, there are several sub-categories. Users can simply choose a broad category (e.g. **720000 Accommodation and Food Services**) or choose a specific sub-category (e.g. **722110 Full-Service Restaurants)** within that category.
4. When you have selected the appropriate code, select **OK**. The dialog box will close and the NAICS code and title will appear in the **Attributes** tab.

Under **Construction Type**, enter information about the construction style, quality, materials, etc. of a structure.

FIA will use the data entered in the **Attributes** tab to compute the value of structures. At the bottom of the **Attributes** tab, select whether FIA should compute structure value using price per square foot (recommended) or using an algorithm.

8.9   Delete a Structure Inventory

To delete a structure inventory in HEC-FIA: Starting in the **Study Pane**, right-click on **Global Inventory**. From the shortcut menu, select **Delete All Structures** (Figure 8.23).
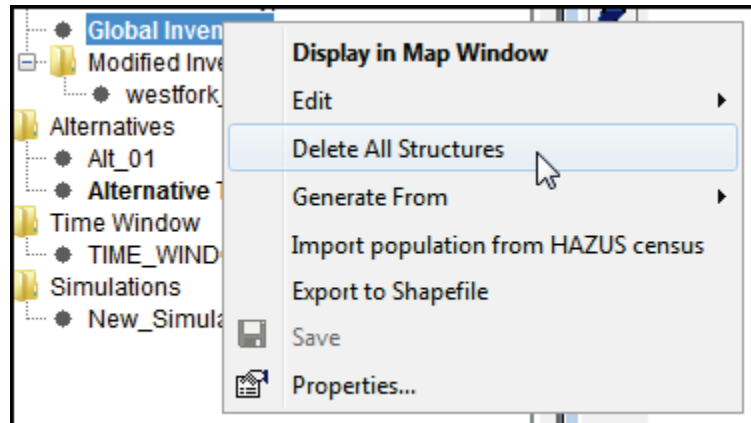
Figure 8.23 Global Inventory shortcut menu (Study Pane).

A warning box will appear, prompting the user to confirm deletion. To delete, select **Yes** (Figure 8.24).
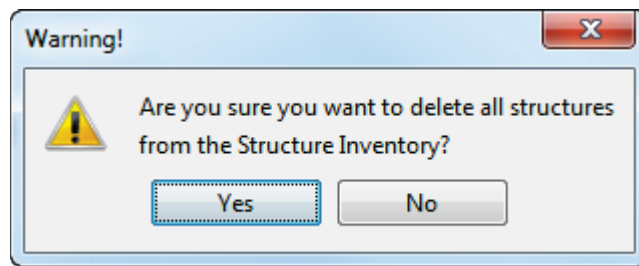
Figure 8.24 Warning! dialog box.

8.10   Edit Structure Occupancy Types

In the previous section 8.4 we learned how to assign a **Structure Occupancy Type** to a structure. In this section we will learn how to create and edit a **Structure Occupancy Type.**

A **Structure Occupancy Type** defines the **Structure Depth-Damage Function** that HEC-FIA will use to compute structure damages. Each occupancy type is assigned a code that indicates its zoning (commercial, residential, or industrial) and construction type. Table 8.1 shows the names and descriptions of occupancy types utilized with the HAZUS data import process in FIA along with the damage category each belongs to.

Table 8.1 HAZUS Occupancy Type Names and Definitions.

Damage Category	Occupancy Type	Description
Residential	RES1-1SNB	Single family home, 1 story no basement
	RES1-1SWB	Single family home, 1 story, basement
	RES1-2SNB	Single family home, 2 story no basement
	RES1-2SWB	Single family home, 2 story, basement
	RES1-3SNB	Single family home, 3 story no basement
	RES1-3SWB	Single family home, 3 story, basement
	RES1-SLNB	Single family home, split-level, no basement
	RES1-SLWB	Single family home, split-level, basement
	RES2	Mobile home
	RES3AI	Multi-Family Dwelling: Duplex
	RES3BI	Multi-Family Dwelling: 3-4Units
	RES3CI	Multi-Family Dwelling: 5-9Units
	RES3DI	Multi-Family Dwelling: 10-19Units
	RES3EI	Multi-Family Dwelling: 20-49Units
	RES3FI	Multi-Family Dwelling: 50+Units
	RES4	Hotel or Motel
	RES5	Institutional Dormitory (Group Housing/Jails)
	RES6	Nursing Home
Commercial	COM1	Retail trade.
	COM2	Wholesale Trade (Warehouse)
	COM3	Personal/Repair Service
	COM4	Professional/Technological Service (Office)
	COM5	Bank
	COM6	Hospital
	COM7	Medical Office/Clinic
	COM8	Entertainment and Recreation (Restaurants/Bars)
	COM9	Theater
	COM10	Parking garage
Industrial	IND1	Heavy industrial
	IND2	Light industrial
	IND3	Food/Drug/Chemicals
	IND4	Metal/Mineral Processing
	IND5	High Technology
	IND6	Construction (Office)
Commercial	AGR1	Agriculture
	REL1	Church or Non-Profit
Public	GOV1	General Government Services (Office)
	GOV2	Emergency Response
	EDU1	Primary/Secondary School
	EDU2	College/university

8.11   Create a New Structure Occupancy Type

To create a new **Structure Occupancy Type**:

1. In the **Study Pane**, under the **Structure Inventory** sub-folder right-click on **Structure Occ. Types**. From the shortcut menu, select **Edit → Form Editor** (Figure 8.25).
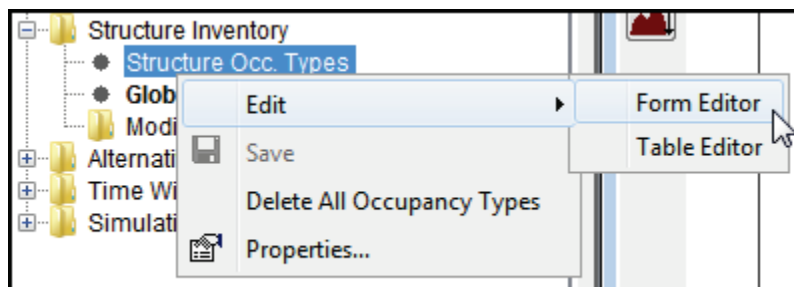
Figure 8.25 Structure Occupancy Type shortcut menu (Study Pane).

2. The **Structure Occupancy Type Editor** will appear (Figure 8.26).
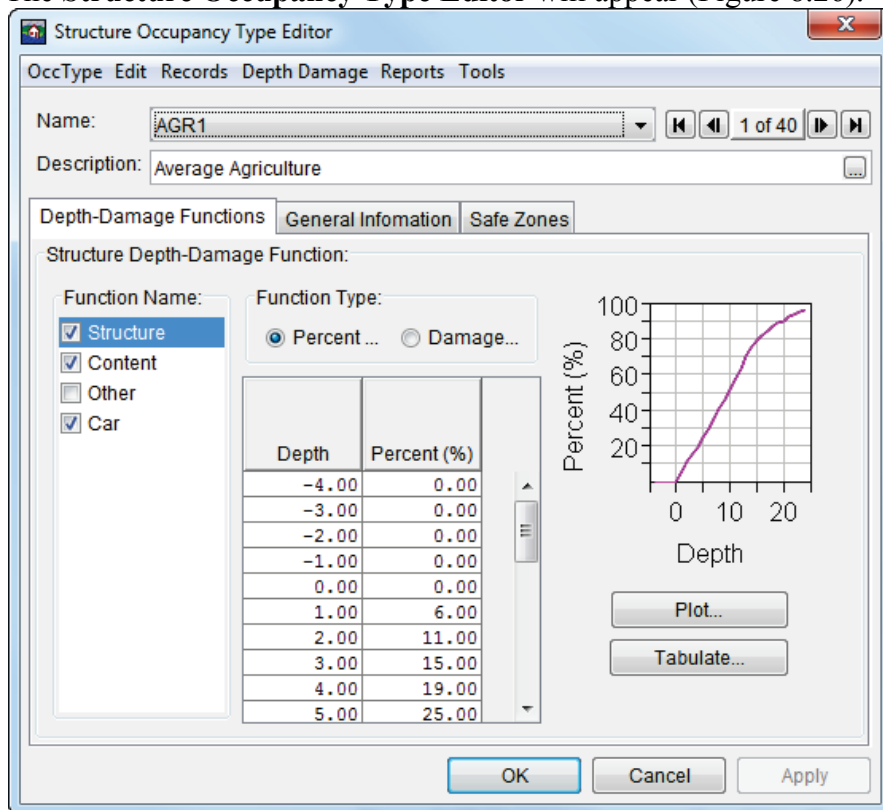
Figure 8.26 Structure Occupancy Type Editor.

3. From the **OccType** menu, select **New**. A dialog box will prompt you to enter in a name (required) and a description (Figure 8.27). Press **OK** to create a new Structure Occupancy Type.
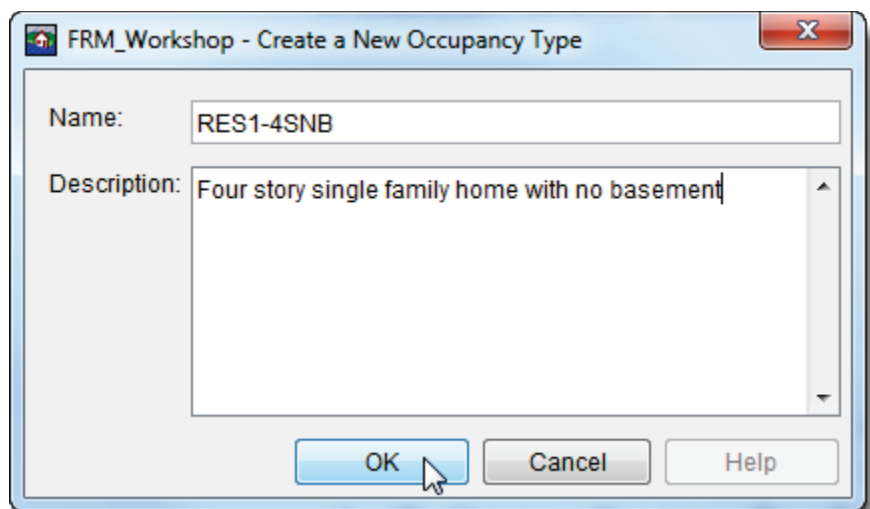
Figure 8.27 Create a New Occupancy Type dialog box.

3. The dialog will close and you will be taken back to the **Structure Occupancy Type Editor**. The Structure Occupancy Type you have just created will be selected.

Now you will have to define the damage function associated with the occupancy type as well as data used for calculating economic and life-loss damage. These are entered using the three tabs of the **Structure Occupancy Type Editor**: **Depth-Damage Functions**, **General Information**, and **Safe Zones**.

8.11.1   Depth-Damage Functions Tab

The **Depth-Damage Functions** tab (Figure 8.26) will be open by default. In the table, enter the numbers that represent a depth-damage function for the occupancy type. The function can be defined either as inundation depth to percentage of structure damaged or inundation depth to dollars of damage.

Included in a Structure Occupancy Type are up to four different categories of damage:
- **Structure** – damage to the structure itself
- **Content** – damage to the contents of a structure
- **Other** – a user-defined category of damage not included in the three other categories
- **Car** – damage to a vehicle associated with the structure

Each category can be selected for editing in the window labeled Function Name to the left of the spreadsheet (see Figure 8.17).

Load Depth-Damage Function from Hazus Data

In a **Structure Inventory** generated from HAZUS data (see Section 8.1.1), HEC-FIA allows users to load damage functions from HAZUS data.

To load a damage function from HAZUS data:

1. Open the **Structure Occupancy Type Editor** (Figure 8.25). From the **Name** menu, select the name of the **Structure Occupancy Type** to which you wish to assign a damage function.
2. From the **Menu Bar**, select **Depth Damage⟶Load Hazus Damage Function** (Figure 8.28).
3. The **Hazus Damage Functions** dialog box will appear, prompting you to select Occupancy Type, data source, and a description (Figure 8.29). Select the appropriate items and click **OK**.
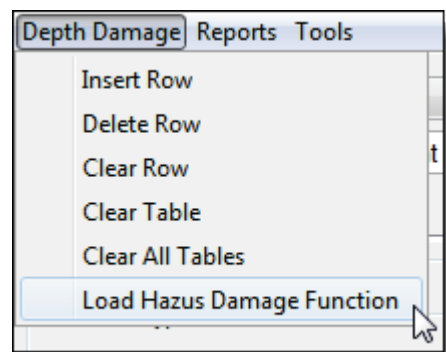
Figure 8.28 Depth Damage menu (Structure Occupancy Type Editor).
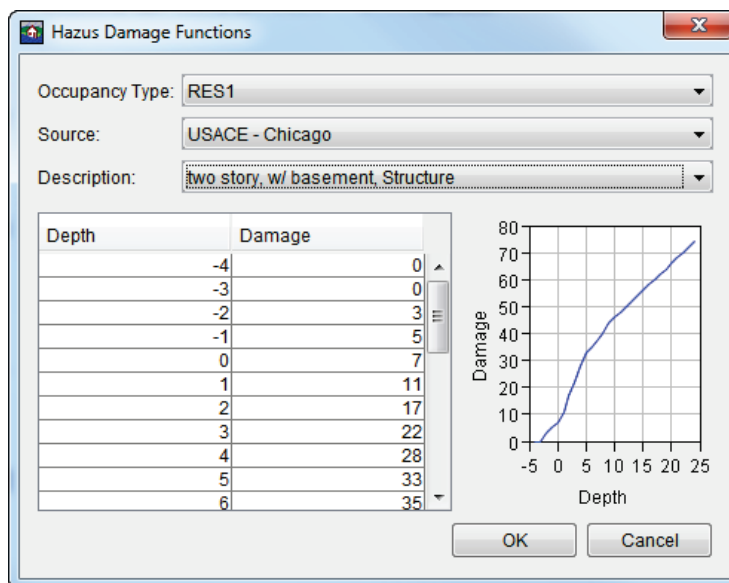
Figure 8.29 Hazus Damage Functions dialog box

4. To accept the changes, save or click Apply.

8.11.2   General Information Tab

Under the **General Information** tab, enter additional data in a variety of categories you would like to associate with a **Structure Occupancy Type** (Figure 8.30). The data entered in the structure occupancy type will overwrite data entered on an individual structure level. On the other hand, once data has been globally assigned in the occupancy type editor, it can then be edited on an individual structure level through the structure editor. It is up to the user to decide which fields should be globally edited for a given occupancy type. For example, in a **Structure Occupancy Type** for four-story residences with no basement (*RES1-4SNB*), all of the structures should have four stories. The value, foundation height, car value, etc. will vary depending on the structure.

Figure 8.30 Structure Occupancy Type Editor—General Information tab.

8.11.3   Safe Zones

FIA also uses parameters pertaining to fatality rates in different levels of a structure during a flood event to calculate life loss. These are calculated according to user-editable **safe zones** within a structure defined in the **Safe Zone** tab of the **Structure Occupancy Type Editor** (Figure 8.31).

A **safe zone** is a portion of a structure level for which users can define the likelihood that an occupant will escape to safety. Since it may be more difficult for a person over 65 to reach safety than for a person under 65, users can define separate **safe zone** boundaries for persons under or over 65. This functionality helps FIA estimate population over 65 at risk during a flood event. Later, when users create alternatives, they will enter fatality rate statistics for each of the **safe zones**.
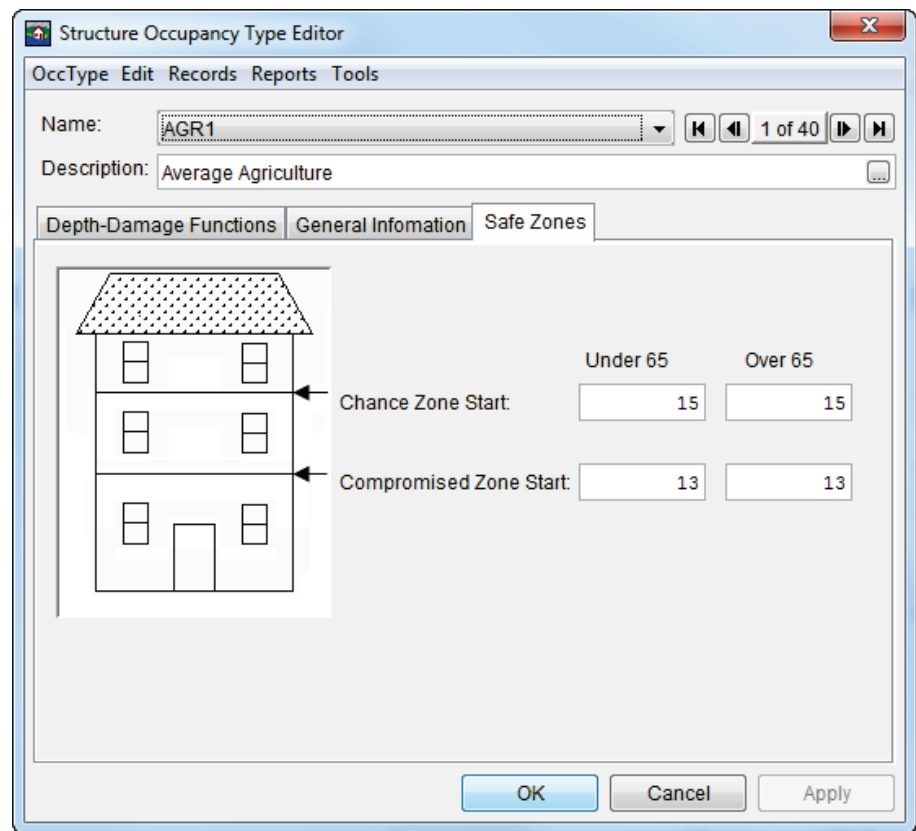
Figure 8.31 Structure Occupancy Type Editor—Safe Zones tab.

Within each of these structures are three user-definable safe zones for which separate fatality rates can be entered. The risk zones are defined below.

    a. The **Safe Zone** is the safest place to be in a flood event. Its boundaries are defined as the area between the foundation of the structure and the **Compromised Zone Start**.

    b. The **Compromised Zone** is an area where an individual's ability to evacuate is doubtful. Its boundaries are defined as the area between **Compromised Zone Start** and **Chance Zone Start**.

    c. The **Chance Zone** is the most risky place to be caught in a flood event. Its boundaries are defined as the area above the **Chance Zone Start**.

Define the boundaries of these safe zones by entering values in the **Chance Zone Start (Under/Over 65)** and **Compromised Zone Start (Under/Over65)** fields.

8.12   Edit an Existing Structure Occupancy Type

To edit a **Structure Occupancy Type** in HEC-FIA:
1.   From the **Name** list, select the name of the occupancy type you wish to edit (Figure 8.32).
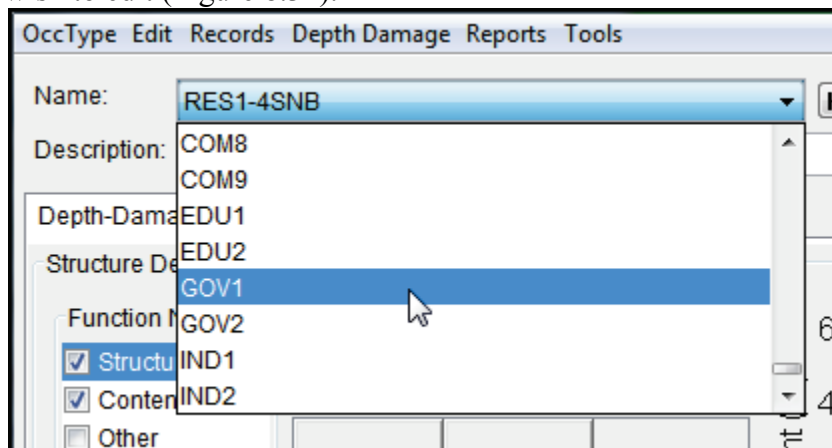
Figure 8.32 Name list (Structure Occupancy Type Editor).

2.   The **Depth-Damage Function** and **General Information** tabs will now be available for editing. Change the information as desired and click <u>Apply</u>.

When users browse to a different a structure occupancy type, HEC-FIA will always prompt them to save their changes.

8.13   Delete a Structure Occupancy Type

To delete a **Structure Occupancy Type**:
1.   Open the **Structure Occupancy Type** editor (Figure 8.25).
2.   From the Names dropdown menu (see Figure 8.32), select the name of the **Structure Occupancy Type** you wish to delete.
3.   Open the **OccType** dropdown in the **Menu Bar** and select **Delete**. Alternatively, simply press Ctrl+D on your keyboard.
4.   A dialog box will appear asking you to confirm deletion. To delete, select **Yes** (Figure 8.33).
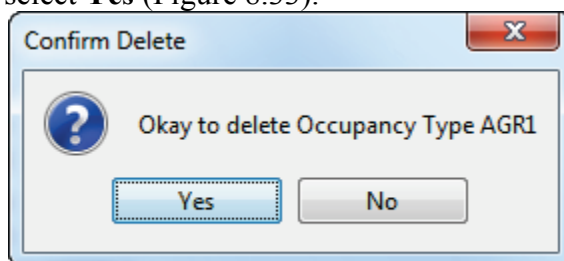
Figure 8.33 Confirm Delete dialog box.

8.14   Modified Inventory

In order to assess damages with modifications to structures (i.e., raising foundation heights, moving structures altogether, etc.), HEC-FIA allows users to create a **Modified Inventory**. This would be the primary place that users would evaluate the effectiveness of non structural measures applied to the structural inventory.

8.14.1   Create a Modified Inventory

To create a **Modified Inventory** in FIA:
1. In the Study Pane, under Structure Inventory, right click on the Modified Inventory subfolder and select **New** (Figure 8.34).
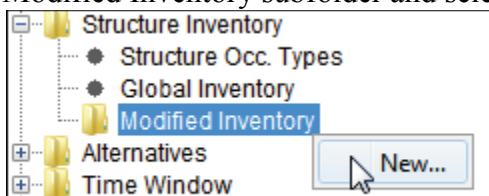
Figure 8.34 Modified Inventory folder shortcut menu (Study Pane).

2. The **Create New Modified Inventory** dialog box will appear. Enter a name and a description and select **OK** (Figure 8.35).
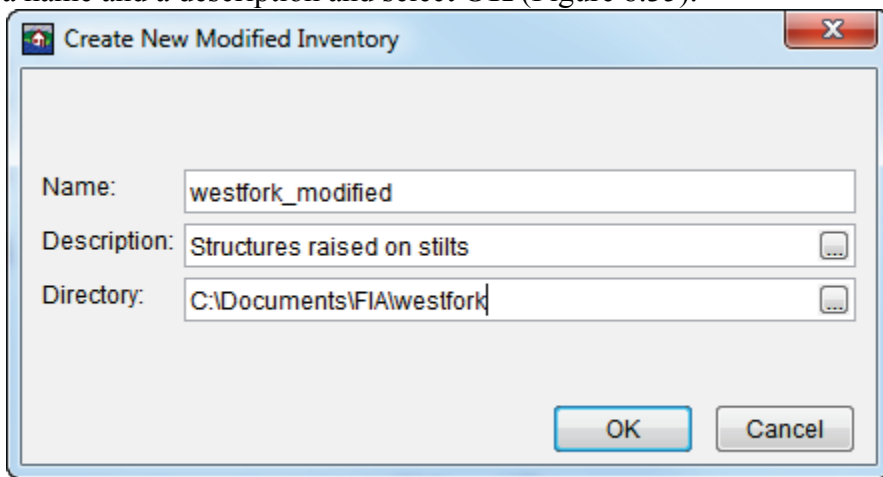
Figure 8.35 Create New Modified Inventory dialog box.

3. The name of your new **Modified Inventory** will now appear in the Study Pane under the **Modified Inventory** subfolder and the modified structure inventory will be displayed in the map window.

8.14.2   Edit a Modified Inventory

Editing a **Modified Inventory** follows the same process as editing the global **Structure Inventory**. Create new **Structure Occupancy Types**, if desired, in the **Structure Occupancy Type** editor (see Section 8.10) and

then assign them to your **Modified Inventory** in the **Structure Editor** for your Modified Inventory.

The functions of the **Modified Inventory Structure Editor** are covered in detail in **Section 8.2**. To access the **Modified Inventory Structure Editor**, right click on the filename of the modified inventory you wish to edit. From the shortcut menu, select **Edit ⟶Form Editor**.
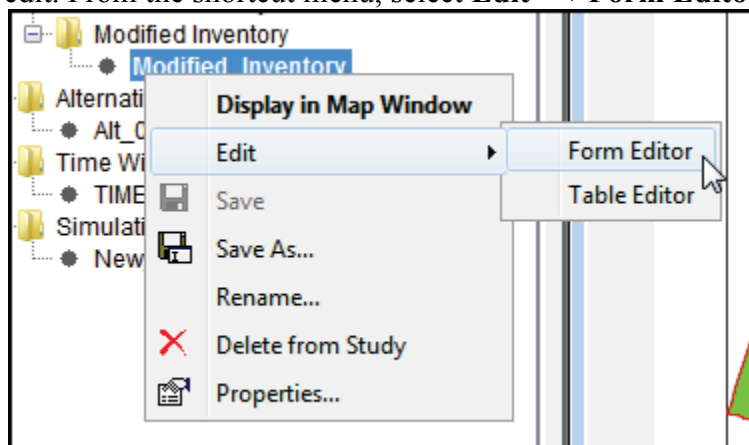
Figure 8.36 Modified Inventory shortcut menu (Study Pane).

8.14.3   Delete a Modified Inventory

To delete a **Modified Inventory**, locate the filename of the inventory you wish to delete (e.g. *westfork_modified*) and right-click. From the shortcut menu, select **Delete From Study** (Figure 8.36). HEC-FIA will ask you to confirm deletion in a dialog box. The files associated with the inventory will also be deleted from your hard drive. To delete, select **Yes**.

Chapter 9

Alternatives

Once users have entered all the required data for their study, they are ready to create alternatives. An **alternative** in HEC-FIA defines what type of simulation will be computed and links the impact area attributes to the simulation. In an alternative, a user defines the simulation as either a gridded compute or a DSS time-series data compute. Multiple alternatives can be created each with their own unique characteristics.

9.1   Alternative Concepts

To create an alternative, an impact area system has to have been created for a specific configuration. For example, if you were studying the effects of a proposed upstream flood-control reservoir, you might create two alternatives: a with-reservoir alternative and a without-reservoir alternative. The without-reservoir alternative would consist of DSS records or grids that were developed from the existing stream gages in the study area, and an impact area system relating the economic conditions of the study area. The with-reservoir alternative would consist of DSS records or grids that were generated from a computer model set-up to simulate the effects of the new reservoir, and a different impact area system with the new reservoir identified.

Setting up multiple alternatives allows you to quantify and later compare the effects or operational changes to a flood-control system. Typically, FIA alternatives fall in one of two classes: with-project or without-project. For example, if you were studying the effects of a levee, the without-project alternative would be the conditions of the system before the levee is built, and the with-project alternative would be the estimated conditions of the system after the levee is built.

There are two ways to set up alternatives in HEC-FIA: through gridded data or time-series HEC-DSS hydrographs.

9.1.2   Inundation Grid Data Concepts

Inundation grid data is used by HEC-FIA in the analysis of flood damage. FIA uses grid data in the form of depth grids, arrival time grids, and duration grids. Depth grids are used to analyze water depth at structures and agricultural areas for damage computations. Arrival time grids are

used to analyze when water arrives for use in the life loss computation and agricultural damage computations. Duration grids are used in agricultural damage computations.

Grid data is defined as rows and columns of cells that are equally spaced. Each grid cell is actually defined by a point at the cell center. Each cell has an attribute or set of attributes representing the area of that cell. The attributes of a cell for use in FIA are depth, ground elevation, and time. FIA accepts grid data in two formats; .FLT (float) and .smrmp (Segmented Multi-Resolution Multi-Parameter). Float files can be generated directly from HEC-RAS Mapper or converted to float format through ESRI software. smrmp files are FIA specific and can be generated through the FIA interface.

9.1.3   Hydrograph Time-series Data Concepts

HEC-FIA can also use hydrograph time-series data (stage or discharge) in the analysis of flood damage. To simplify managing time-series, FIA uses the concept of a hydrograph time-series set.

The impact of flooding in urban areas is a function of peak stage. That is, as a flood stage rises and then falls in an urban area, the damage incurred depends primarily on the maximum depth reached. In fact, the urban damage incurred during a year is well modeled as a function of annual maximum stage. This relationship is so because the reconstruction period for residential, industrial, commercial, and public structures and their contents is typically about one year.

Estimating crop damage is more complicated. Flood damage to crops depends upon the season during which the flood occurs, the duration of flooding, and the dryout period between the floods. To account properly for these contributing factors, FIA continuously simulates conditions in an impact area during a flood, noting the season, computing the duration, and identifying the dry-out period.

To permit this continuous simulation, a time-series of stage for the impact area must be provided. In the simplest case, which is illustrated by Figure 9.1(*a*), you can provide a stage time-series for a gage at or adjacent to the impact area, and FIA will use this for the damage computations. The hydrograph can be of any duration, and, for analysis in FIA, the ordinates can be specified at any of the following intervals: 1, 2, 3, 4, 6, 12, and 24 hours.

As an alternative to providing a stage hydrograph, you can provide a flow hydrograph at or adjacent to the impact area, along with a rating function, as illustrated by Figure 9.1(*b*). The rating function relates flow to stage.
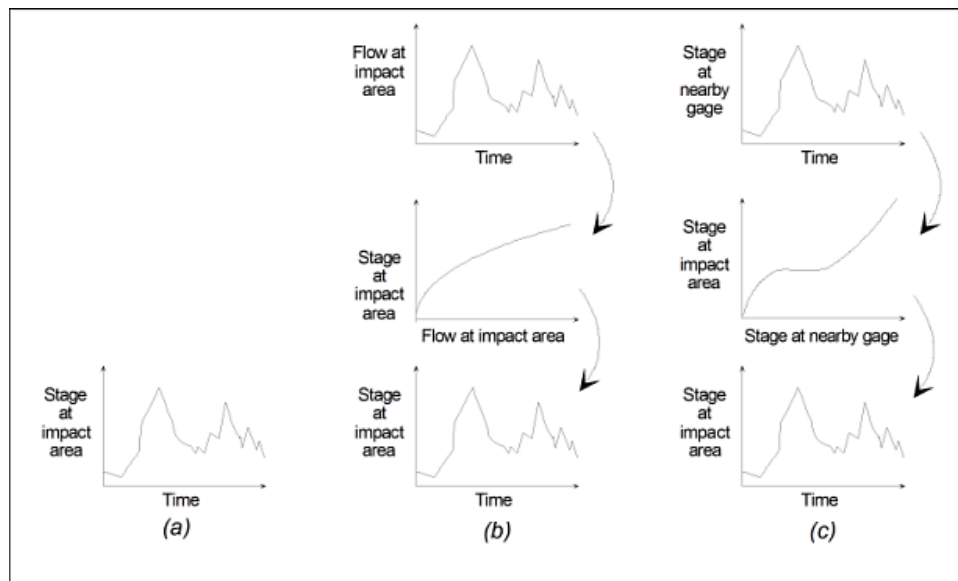
Figure 9.1 Illustration of Time-series Alternatives

FIA can transform the flow hydrograph time-series to a stage hydrograph time-series.

Finally, if you do not have access to stage at a gage adjacent to the impact area of interest, you can use stage data from a nearby gage. Use an independent hydraulics model to allow FIA to determine how the stage at the impact area relates to the stage at the gage. As illustrated by Figure 9.1(*c*), the hydraulics model predicts stage at the impact area as a function of stage at the gage. FIA uses the transfer table, described in detail in Section 9.7.4, to transfer the stage data.

9.1.4   HEC Data Storage System Concepts

The hydrograph time-series data required for impact analysis are stored in an HEC Data Storage System (HEC-DSS) database. DSS is a database management system that is designed specifically to store data for applications in water resources. DSS can store many types of information, but it is most efficient at storing time-series. The data are arranged into blocks, for example one year of daily-interval data, is stored in a single **record** in the DSS. Each record is given a unique name called a pathname. The **pathname** is formulated in a way that makes it easy to group related data together.

Software from HEC has the ability to read from and write to DSS. This capability facilitates data exchange among applications. For example, you could use HEC-ResSim to generate the time-series input for FIA. This application stores time-series data in DSS. Other applications from HEC

permit data entry, conversion to and from other databases, plotting, tabulating, and organizing a multitude of data sources and values.

The pathname of a DSS record can be three hundred and ninety-one (391) characters or less, and is, by convention, separated into **six parts**. The parts are delimited by slashes in the pathname. The pathname parts are designated **A** through **F**. Additional conventions hint at the contents of the record that each pathname part represents. So, for the pathname:

```
/BALD EAGLE/SUSQUEHANNA/FLOW-REG/01NOV1993/1HOUR/OBS
```

Table 9.1 defines the various parts of the pathname. The C-Part usually follows a set convention and describes the type of data. The F-Part is usually used to distinguish one time-series from another. In the example (Table 9.1) the time-series is observed data; another possible type might be computed.

Table 9.1 HEC-DSS Pathname Parts

A	BALD EAGLE	name of watershed
B	SUSQUEHANNA	hydrograph location
C	FLOW-REG	type of data; regulated flow discharge
D	01NOV1993	starting date of the block of data in the record
E	1HOUR	the data are spaced in time at one-hour intervals
F	OBS	observed data

9.2   Create Alternative from Gridded Data

To create a new alternative from gridded data:

1. Starting in the **Study Pane**, right-click on the **Alternatives** folder. From the shortcut, select **New** (Figure 9.2).
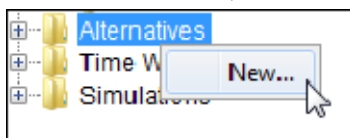
Figure 9.2 Alternatives Folder shortcut menu (Study Pane).

2. The **Create New Alternatives** dialog box will open (Figure 9.3). Enter a name (required) and a short description. Using the ⊡ browse button, select the directory in which your new alternative should be created.

3. From the dropdown lists, select the **Project Configuration**, **Structure Inventory**, and **Agricultural Grids** you would like to associate with your new alternative.

4. At the bottom of the dialog box, **be sure to select Inundated area grid** (Figure 9.3).
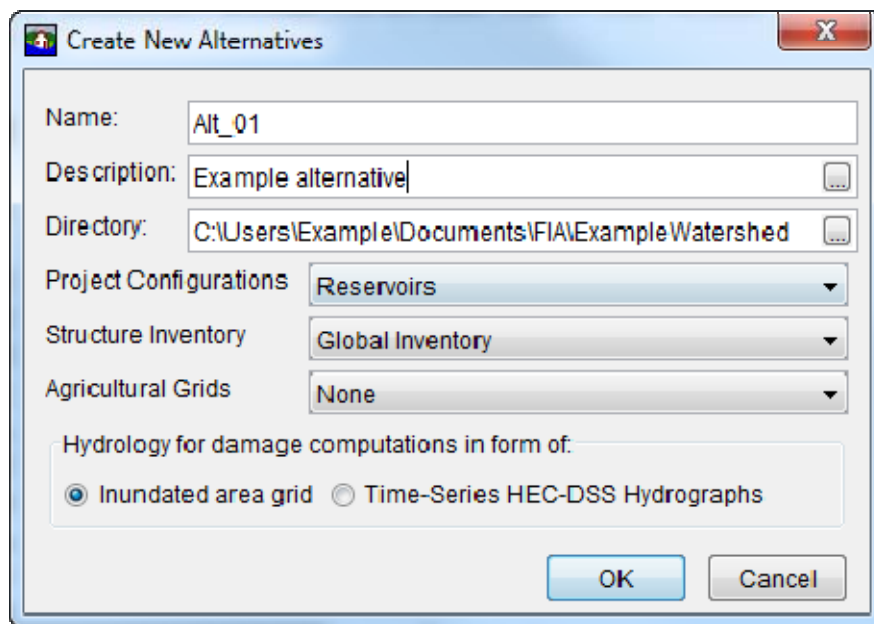
Figure 9.3 Create New Alternatives dialog box.

5.   Once all information has been defined, select **OK**.
6.   Your new alternative will now appear in the **Study Pane** under the **Alternatives** folder.

9.3   Create Alternative from DSS Hydrograph

To create a new alternative from time-series DSS hydrograph data:

1.   Starting in the **Study Pane**, right-click on the **Alternatives** folder. From the shortcut, select **New** (**Error! Reference source not found.**).
2.   The **Create New Alternatives** dialog box will open. Enter a name (required) and a short description. Using the 🔲 browse button, select the directory in which your new alternative should be created.
3.   From the dropdown lists, select the **Project Configuration**, **Structure Inventory**, and **Agricultural Grids** you would like to associate with your new alternative.
4.   At the bottom of the dialog box, **be sure to select Time-Series HEC-DSS Hydrographs** (Figure 9.4).
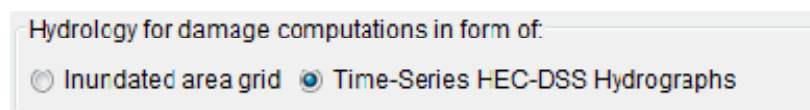
Figure 9.4 Create alternative from time-series hydrographs.

5.   Once all information has been defined, select **OK**.
6.   Your new alternative will now appear in the **Study Pane** under the **Alternatives** folder.

9.4   Save Alternative Data

Whenever unsaved changes are made to an alternative, an asterisk will appear next to the alternative name in the **Study Pane** (e.g. *Emergency_Notification**). To save these changes:

1. Starting in the **Study Pane**, under the **Alternatives** folder locate the name of the unsaved***** alternative you wish to save. From the shortcut menu, select **Save** (Figure 9.5).
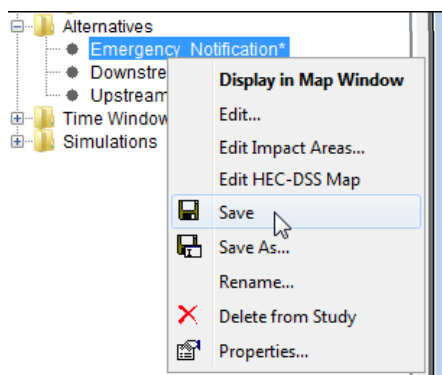
Figure 9.5 Alternatives shortcut menu (Study Pane).

2. The program will automatically save all data associated with the alternative.

9.5   Save As

> *To create a copy of an existing alternative, use the **Save As** command from the **Alternative Shortcut Menu**.*

The **Save As** feature allows you to create a copy of an alternative with a different name. This feature is useful if you want to create two similar alternatives, but with different time-series mapping. Instead of creating two original alternatives from scratch, users can create and populate one alternative, then use the **Save As** command to create a new alternative. This new alternative will be associated with the same configuration as the original alternative.

To create an alternative from an alternative for a particular configuration:

1. Starting in the **Study Pane**, under the **Alternatives** folder right-click the name of the alternative you wish to make a copy of. From the shortcut menu, select **Save As** (see Figure 9.5).
2. The **Save Alternative As** dialog box will appear. Enter a new name in the **Name** box and enter a description in the **Description** box (Figure 9.6). Then define the directory in which HEC-FIA should save the copy of the alternative.
3. Click **OK**. The **Save Alternative As** dialog box will close and the name of the new alternative will appear under the **Alternatives** folder of the **Study Pane**.
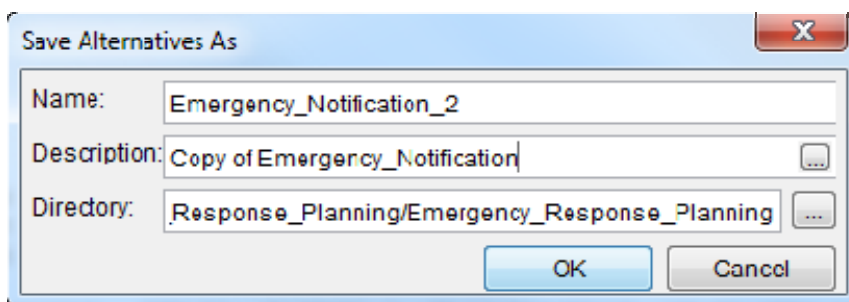
Figure 9.6 Save Alternative As dialog box.

9.6   Delete an Alternative

To delete an alternative:

1. Starting in the **Study Pane**, under the **Alternatives** folder right-click the name of the alternative you wish delete. From the shortcut menu, select **Delete from Study** (option in Figure 9.5).
2. A warning message will open asking you whether you really want to delete the selected alternative. To delete, click **Yes**. The alternative and the files associated with it will be deleted from your hard drive.

9.7   Edit an Alternative by Impact Area

> *For **Time-Series-Based Alternatives,** definition of an Impact Area using the Impact Area Editor is required for damages calculation.*

FIA is designed to evaluate flood damage and other impacts and then sort and categorize these damages and impacts by impact areas. An **impact area** is any distinct portion of a watershed that is affected by a rising stage in a stream, river, lake, or reservoir. Using the **Impact Area Editor**, users characterize the effects of the rising stage by describing the property and population at risk in the area bounded by an **Impact Area** polygon. Depending on whether an **Alternative** is created from **DSS Hydrographs** (Section 9.1.3, page 9-5), or from **Gridded Data** (Section 9.1.2, page 9-4), the way that users describe structure, crop, and population information will be very different.

For **Alternatives Created from Time-Series Hydrographs**, users will enter all the information required for damages calculation into the **Impact Area Editor**. Recall that in the **Global Data Editor**, you entered general information required to define each impact area (see Chapter 4). This information included boundaries, crops, damage categories, and reservoirs. In the **Impact Area Editor**, users apply these properties to each impact area. For example, as shown in Figure 9.7, you previously defined the various crops in the **Global Data Editor**. Now under the **Crops** tab for an impact area, the table automatically displays the crop information defined in the **Global Data Editor**. With that table, you would then select the pertinent crops for that particular impact area.

Figure 9.7 Impact Area Editor - Crops tab.

With a time-series-based alternative, users can manually enter **Depth-Damage Functions** to compute aggregated damages to structures and population or perform damage calculations on a structure by structure basis.

*For **Grid-Based Alternatives,** damages are calculated from information contained in gridded data and the FIA structure inventory and hence some features of the Impact Area are grayed out. However, **Life-Loss Data** and a few other functions must be entered through the **Impact Area Editor**.*

In contrast, for an **Alternative Created from Gridded Data**, since the **Inundated-Area**, **Agricultural**, and **Structure Inventory** data themselves contain the information used for damage computation, these functions of the **Impact Area Editor** are no longer necessary and thus grayed out (Figure 9.8).

Figure 9.8 For grid-based alternatives, the unnecessary functions of the Impact Area Editor are grayed out

However, much of the functionality of the **Impact Area Editor** is still pertinent to grid-based simulations. Significantly, **Life-Loss Simulation** data is entered exclusively in the **Impact Area Editor**. This functionality is new to HEC-FIA version 2.1. The following sections will describe the functions of each of the nine tabs of the **Impact Area Editor**.

9.7.1   Impact Area Editor

Once an impact area is uploaded (Chapter 5, Section 5.2.1), users can define the properties for each impact area in a study. This is accomplished by using the **Impact Area Editor** (Figure 9.9).

Figure 9.9 Impact Area Editor.

To access the **Impact Area Editor**:

1. Starting in the **Study Pane**, under the **Alternatives** subfolder, right-click the name of the alternative for which you wish to enter impact area data.
2. From the shortcut menu (Figure 9.5), select **Edit Impact Areas**.
3. The **Impact Area Editor** (Figure 9.9) will open.

The remainder of this chapter describes how to enter the data for the individual tabs – **Boundaries, Projects, Hydrograph, Damage Functions, Crops, Adjusted Price Index, Impact Table,** and **Recession Table**.

9.7.2   Boundaries Tab

The **Boundaries** tab (Figure 9.10) allows users to specify boundaries for each impact area for viewing and sorting results. The name of the selected impact area is shown at the top of the tab in the **Impact Area Name** list. To view data for a different impact area, select its name from the **Impact Area Name** list.

The Boundaries tab is where users identify the boundaries that are associated with an active impact area.

For each boundary type, select from the list the appropriate item that defines the boundaries of the active impact area. The boundary information that is available here was entered in the **Global Data Editor** (Chapter 4, Section 4.3).

To select boundary data:

1. You must be in the **Impact Area Editor**. See Section 9.7.1 (page 9-8) on how to access the **Impact Area Editor.**
2. Click the **Boundaries** tab.

Figure 9.10 Impact Area Editor—Boundaries tab.

3. In Figure 9.11 a state boundary is being selected for the active impact area by clicking the appropriate state from the **State** list. That boundary will now be assigned to the selected impact area.

Figure 9.11 State List (Impact Area Editor—Boundaries tab).

4. Repeat step 4 for all of the boundary types that are needed for the selected impact area.

5. Click **Apply** when finished assigning boundaries to impact areas. To exit the **Impact Area Editor**, click **OK**.

9.7.3   Projects Tab

The **Projects** tab (Figure 9.12) is where information is entered about local (levees) and remote (reservoirs) **projects** – man-made structures built to reduce flood damage. FIA allows users to add reservoirs and levees to calculate the benefit the project brings to a watershed area (how much damage is prevented as a result of the levee or reservoir). In this tab, users toggle whether the program should calculate individual project benefits, how the calculations are to be performed, and enter the information necessary to perform the calculations.
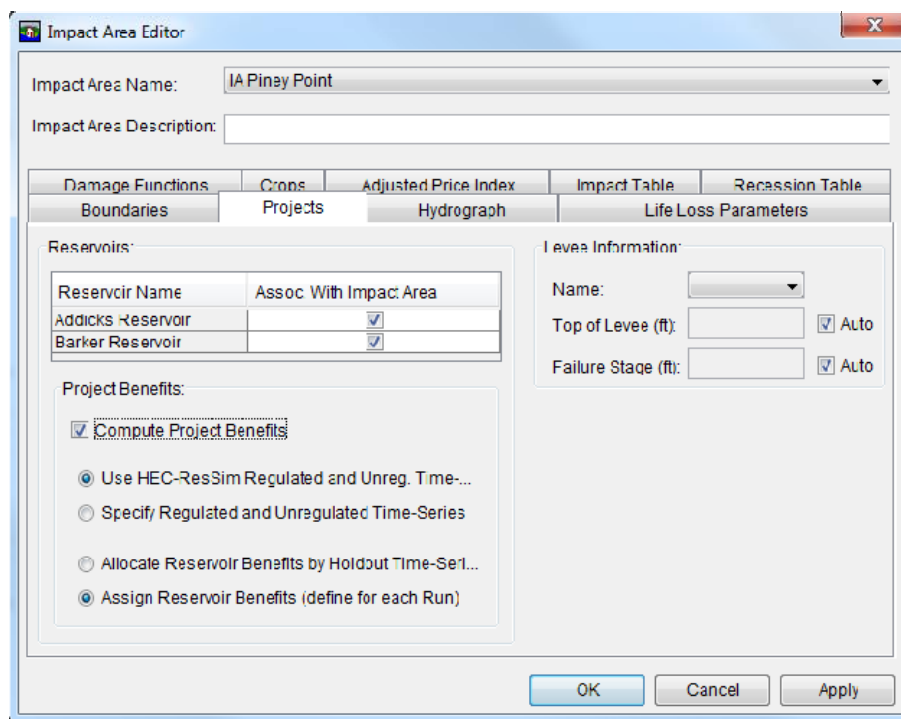
Figure 9.12 Impact Area Editor—Projects tab.

Conceptually within FIA, two categories of projects exist: local and remote. **Local projects** are assumed to modify flood conditions for one impact area. Examples are levees or raising the foundation elevations for a group of homes. Local protection projects are assumed to be functional for the entire impact area and only one impact area. In other words, each impact area protected by a levee must have a different levee name associated with it. If a levee is very long, the levee must be given different names for each impact area it protects.

Remote projects (as defined in FIA) increase or decrease the stage in a river adjacent to one or more impact areas. Building or removing an upstream flood-control reservoir is an example.

Reservoirs

Reservoirs are remote projects, which increase or decrease the stage in a river adjacent to one or more impact areas.

The **Reservoirs** table (Figure 9.13) in the **Projects** tab contains the names of all the reservoirs in your study. From the table of reservoirs you will select the reservoirs upstream of the active impact area that influence the adjacent stage.

To select a reservoir:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).

2. Click **Projects** tab.

3. In the **Reservoir Table**, check the boxes next to the reservoirs you would like to associate with the impact area.

Reservoir Name	Assoc. With Impact Area
Addicks Reservoir	☑
Barker Reservoir	☑

Figure 9.13 Selecting a reservoir (Impact Area Editor—Projects Tab).

4. Click **Apply** when finished assigning reservoirs to impact areas. To exit the **Impact Area Editor**, click **OK**.

FIA automatically calculates the without- and with-project results that are required for allocation of project benefits. These values are required for reporting the annual project accomplishments to Congress, for media presentations following flooding events and for project justification and possible maintenance considerations. This process gives reservoirs initial credit and levees receive any remaining benefits. A full description of the allocation of benefits is provided in Appendix C.

Project Benefits

Project benefit accomplishment associated with Corps of Engineers projects is determined for the period-of-analysis by subtracting the damage (area flooded or structures flooded) associated with the with-project condition from the without-project condition. Benefits are computed for each impact area, and then summed for the system.

Allocation of project benefits at an impact area is optional. When creating a time-series based alternative, **Compute Project Benefits** is set for calculation by default (Figure 9.14). If you chose not to compute project benefits, click **Compute Project Benefits** and the options

Project Benefits:
☑ Compute Project Benefits
● Use HEC-ResSim Regulated and Unreg. Ti...
○ Specify Regulated and Unregulated Time-S...
○ Allocate Reservoir Benefits by Holdout Time...
● Assign Reservoir Benefits (define for each ...

Figure 9.14 Compute Project Benefits option (Impact Area Editor - Projects tab).

will gray out and **Compute Project Benefits** will not be marked for the active impact area.

When you request that project benefits be calculated, there are two options available: **Use HEC-ResSim Regulated and Unreg. Time-Series** or **Specify Regulated and Unregulated Time-Series**. All of these options have sub-options that are available.

Use HEC-ResSim Regulated and Unreg. Time-Series is the default option. A simulation (run) using ResSim will need to be made. Results (simulation.dss) from the ResSim simulation will need to be linked to the FIA time-series at the hydrograph locations. With the sub-option **Allocate Reservoir Benefits by Holdout Time-Series** clicked (Figure 9.14), FIA then will automatically

calculate the without project (unregulated) and with project (regulated) results. Then based on reservoir holdouts, it will calculate the distribution of benefits between the appropriate reservoirs. If there is a levee, FIA will calculate damage for the following conditions:

> With Levee (regulated)
> Without Levee (regulated)
> Without Levee (unregulated)

Specify Regulated and Unregulated Time-Series (Figure 9.15) is the second option. When this option is selected, users must provide the unregulated and regulated hydrographs for calculation of project benefits. The only sub-option available is the **Assign Reservoir Benefits (define for each Run)**. Users will need to manually allocate project benefits to each reservoir, for each impact area, as a percentage. See Section 9.8.2 for manual reservoir benefit allocation.
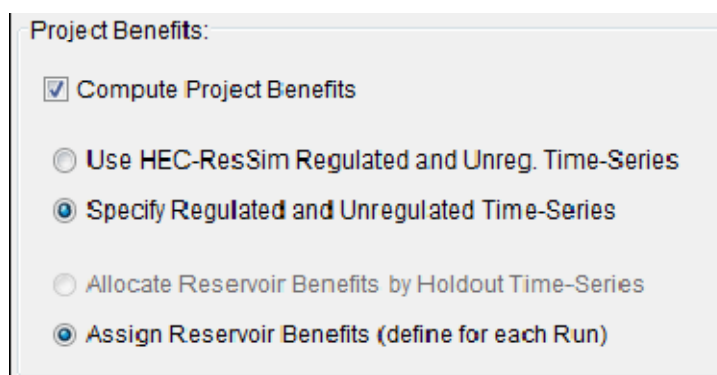
Figure 9.15 Compute Project Benefits using the Specify Regulated and Unregulated Time-Series option (Impact Area Editor - Projects Tab).

See Appendix C for more details on reservoir benefit allocation.

Levee Information

Only one levee project may exist for an impact area. When levee information is entered for an impact area, FIA assumes that all damageable property is protected by the levee, so no damage is computed for that impact area unless the water stage adjacent to that impact area exceeds the levee failure stage. If you do not enter a value for failure stage, FIA will assume that no flood damage in the impact area will occur, until the levee is overtopped (stage exceeds top of levee). See Chapter 5, Section 5.4 for creating and editing levees.

Definition of data entry items follows:

- **Name** – name of a given levee.
- **Top of Levee** – stage at the top of the levee, floodwall or tidal barrier.
- **Failure Stage** – stage below top of levee which causes failure of the levee.

To define a levee:

> *Levees are local projects that modify flood conditions for one impact area.*

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Select the **Projects** tab.
3. Under **Levee Information**, select a levee name from the **Name** dropdown list. In Figure 9.16, *Barker Levee* has been selected from the **Name** list.

Figure 9.16 Edit levee information (Impact Area Editor—Projects tab).

4. To automatically calculate the values for **Top of Levee**, check the **Auto** box. When the **Auto** box is checked for **Failure Stage**, FIA assumes that the failure stage is equal to levee height.
5. If users have more accurate values for **Top of Levee** and **Failure Stage**, clear the **Auto** box and enter the height values in feet for **Top of Levee** and **Failure Stage** manually.
6. Click **Apply** when finished defining levee information for impact areas. To exit the **Impact Area Editor**, click **OK**.

9.7.4   Hydrograph Tab

Within the **Hydrograph** tab (Figure 9.17), users define the name of the hydrograph location (computation point) and type of hydrograph that will be used at an impact area. There are three types to choose from - stage, discharge (flow), or peak-stage hydrograph. A **hydrograph** defines the stage at the impact area that will give the best representation of the average river or reservoir water levels at the index station.

> This function is only pertinent to **Time-Series-Based Alternatives** (concepts in Section 9.7, page 9-7).

<table>
<tr><td>

In the
Hydrograph
tab, users
identify the
hydrograph
location
(computation
point) and type
of hydrograph
that will be used
at an impact
area.

</td><td>

</td></tr>
</table>
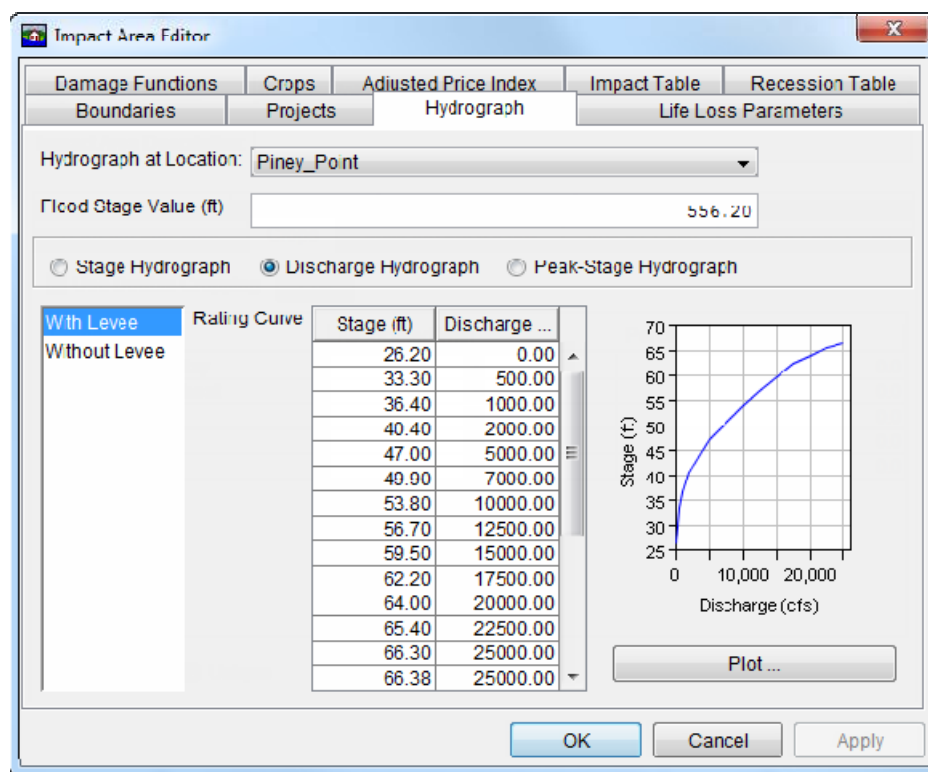
Figure 9.17 Impact Area Editor—Hydrograph tab.

FIA accepts time-series hydrographs, which measure either stage or discharge (flow). However, FIA uses stage values only to compute flood impacts. Thus, if you specify a discharge (flow) hydrograph, you must provide a rating curve so that FIA can convert the discharge to stage. The stage values in the rating curve must be referenced to the same datum as the elevation-damage functions.

FIA allows users to enter a rating curve for with- and without-levee project scenarios. The flow versus stage relationship could be significantly different for these two scenarios and thus the damage reported could be significantly altered with more appropriate rating curves. If a discharge hydrograph is chosen, users can enter a with-levee rating curve, a without-levee rating curve, or both.

If the selected impact area does not have a levee, a without-levee rating curve, at the least, must be entered; if the impact area does have a levee a with-levee rating curve, at the least, must be entered. If both rating curves are entered, then the with-levee rating function is used to compute damage and benefits for the with-levee project and the without-levee rating function is used to compute damage and benefits for the without-levee project. When only entering one rating curve, that rating curve will be used to compute both scenarios.

Definition of data entry items follows:

- **Discharge** – the volume of water passing a specific point for a given time interval
- **Discharge Hydrograph** – a graphical relationship of discharge (flow) as a function of time for a certain location, for FIA purposes a time-series.
- **Flood Stage Value** – highest elevation that has no damage associated with it for the impact area.
- **Hydrograph at Location** – the name of the hydrograph location (computation point) for the impact area (see Chapter 5, Section 5.5).
- **Peak-Stage Hydrograph** – a user-defined hydrograph that requires three points – the peak-stage value plus the peak day and time, beginning event date and time; and, end of event date and time. The beginning event date/time is when the event exceeded the flood stage value of the impact area. The end of event date/time is when the event recedes past the flood stage value of the impact area.
- **Stage** – the vertical distance in feet (meters) above or below a local or national datum (N.G.V.D. for elevations).
- **Stage at Index Location** – index location of the active impact area (stream transfer table).
- **Stage at Reference Location** – location of the actual stage hydrograph (stream transfer table).
- **Stage Hydrograph** – a graphical relationship of stage as a function of time for a certain location, for FIA purposes a time-series.
- **Stream Transfer Table** – for a given impact area the water surface elevations at the hydrograph location may not be the same as the water surface elevations at the index location. To account for this difference a stream transfer relationship is required. Once the relationship is established, FIA will transfer the hydrograph data from the hydrograph location to the index location by accounting for the slope difference in a series of water surface profiles. Intermediate hydrograph values are linearly interpolated.
- **With Levee Rating Curve** – a graphical relationship that yields the stage for a given discharge at a specific location on a stream or river that includes the effects of a levee if present.
- **Without Levee Rating Curve** – a graphical relationship that yields the stage for a given discharge at a specific location on a stream or river that does not include the effects of a levee.

*A **hydrograph** defines the stage at the impact area that will give the best representation of the average river or reservoir water levels at the index*

To define a hydrograph:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Hydrograph** tab.
3. From the **Hydrograph at Location** dropdown list, select the name of the hydrograph you wish to associate with your impact area (Figure 9.18).
4. In the **Flood Stage Value** box, type a number that reflects the flood stage value. For the example in Figure 9.18, *556.20* ft is entered.

5.   Next you must choose the type of hydrograph; there are three options – **Stage Hydrograph, Discharge Hydrograph** (default), or **Peak-Stage Hydrograph**.

Figure 9.18 Hydrograph definition (Impact Area Editor—Hydrograph Tab).

To define a **stage hydrograph**:

a.   Click **Stage Hydrograph**, the **Hydrograph** tab changes. FIA assumes that a stage hydrograph is stored as time-series data; therefore the user will not enter the stage hydrograph directly.

b.   If you do not have a stage hydrograph for the impact area, but you know the relationship between the stage at the impact area index station and the stage at a location at which you have a hydrograph, you can choose the **Use Transfer Table** option.

c.   Click **Use Transfer Table**. The **Stream Transfer** table (Figure 9.19) is now available for data entry.

Figure 9.19 Stream Transfer Table for Stage Hydrographs (Impact Area Editor—Hydrograph tab).

d.   As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot** (option in Figure 9.17, page 9-15).

Define a **discharge hydrograph**:

a.   Click **Discharge Hydrograph** (default), the **Hydrograph** tab changes. FIA assumes that a discharge (flow) hydrograph is stored as time-series data; therefore

the user will not enter the discharge (flow) hydrograph
directly.

b. For a discharge (flow) hydrograph, you must enter a
Rating Curve (Figure 9.17, page 9-15). FIA uses the
rating curve to convert the discharge (flow) values to
stage values for computation.

c. Enter numbers (stage and discharge) that will create the
rating curve using the provided tab. The table will
automatically resize to fit pasted items.

Define a **peak-stage hydrograph**:

a. Click **Peak-Stage Hydrograph**, the **Hydrograph** tab
changes. Under this tab you will find the message, "**A
Peak Stage Hydrograph is Defined for each Run**".
Since peak-stage hydrographs are simulation dependent,
data entry occurs at simulation time.

6. As values are entered, the thumbnail plot to the right of the table
will continually update. To view a full-size plot, click **Plot** (see 17,
page 9-15).

7. Click **Apply** when finished editing hydrograph data for impact
areas. To exit the **Impact Area Editor**, click **OK**.

> *A **peak-stage
> hydrograph** is a
> user defined
> hydrograph that
> requires three
> points – the peak-
> stage value plus the
> peak day and time,
> beginning event
> date and time; and,
> end of event date
> and time.*

9.7.5   Population Adjustment Tab

The **Population Adjustment** tab (Figure 9.20) allows the user to index
population by impact area. The input is in decimals, in that if the user puts
1.1 as seen in Figure 9.20, it would increase the population by ten percent
within the impact area at compute time. Conversely, if the user input .9
the population would be decreased by ten percent within the impact area at
compute time.
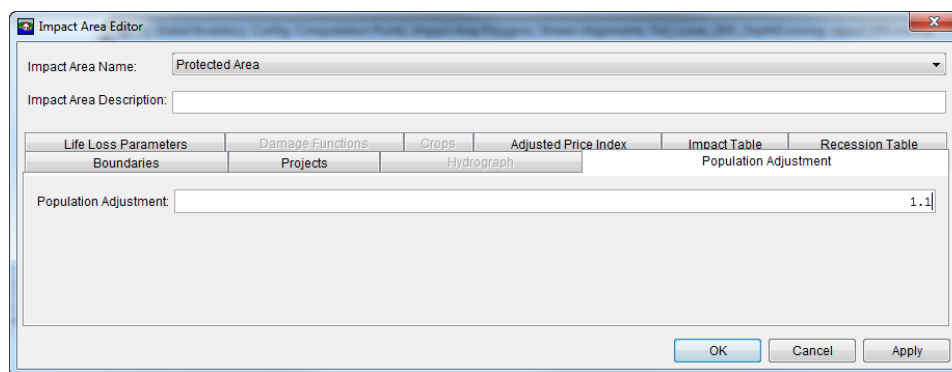
Figure 9.1 Impact Area Editor - Population Adjustment Tab.

9.7.6   Life Loss Parameters Tab

With the **Life Loss Parameters** tab (Figure 9.21), HEC-FIA allows users
to calculate life loss impacts of flood events using curves that represent
Warning Diffusion and **Mobilization** over time while taking into account

Figure 9.21 Impact Area Editor—Life Loss Parameters tab.

user-defined zones within structures. Users can enter different life loss parameters for each impact area or have the information they fill out **Apply to all Impact areas** by checking the box at the top left of the tab. The following sections will explain how to define **Warning Diffusion** and **Mobilization** curves as well as how to modify safe zone parameters within structures.

Warning Diffusion Concepts

HEC-FIA calculates percentage of population warned over time by incorporating *Rogers and Sorensen's* 1991 theorem on diffusion of emergency warnings.

The theorem calculates the rate of change over time of percent of population warned as:

$$\frac{\mathrm{dn}}{\mathrm{dt}} = k\left[a_1 \cdot a_1 f(N-n)\right] + (1-k)\left[a_2 n(N-n)\right]$$

where:

k =   the portion of the population alerted by the warning system who understand its message.

a_1 =   the alerting parameter that defines the efficiency of the broadcast process in reducing uncertainty in a given warning system, i.e. the proportion of unwarned individuals warned during each time step (t_i to t_{i+1}) by the broadcast process (Rogers 7).

a_1f =   multiplier for warning system effectiveness adjusted for the location and activities of individuals throughout the day.

N =   the limit of percentage of population warned within 30 minutes of initial warning time.

n =   the value of N in the preceding time step.

$(1-k)$ =   the portion of the population that cannot be fully alerted or do not fully understand the warning (where 1=the entire population).

a_2 =   multiplier for the effectiveness of the warning system in reducing uncertainty through secondary avenues, i.e. the proportion of unwarned individuals warned during each time step (t_i to t_{i+1}) by secondary means, such as word of mouth (Rogers 7).

FIA allows users to modify a few of these parameters to effectively calculate the diffusion of a warning message over time in a given study.

Define a Warning Diffusion Curve

FIA allows users to utilize one of six preset warning diffusion functions for different warning systems or to define their own warning diffusion curve. One warning diffusion curve can be used for each **Impact Area**.

The preset warning system types are as follows:

- **Sirens**
- **Tone-Alert Radios**
- **Auto-Dial Telephones**.
- **EBS**—Emergency Broadcast System
- **EBS and tone alert radios**—population warned by both the Emergency Broadcast System and tone-alert radios
- **EBS, sirens and auto dial telephones**—population warned by combination of EBS, sirens, and auto-dial telephones

To select a preset function for a warning system type:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. From the **Impact Area Name** dropdown list, select the impact area you wish to edit.
3. Select the **Life Loss Parameters** tab.
4. Locate the dropdown list labeled **Warning System**. From the list, select the warning system used in the impact area you are modifying. In the example (Figure 9.22), the **Tone-Alert Radios** option is selected.
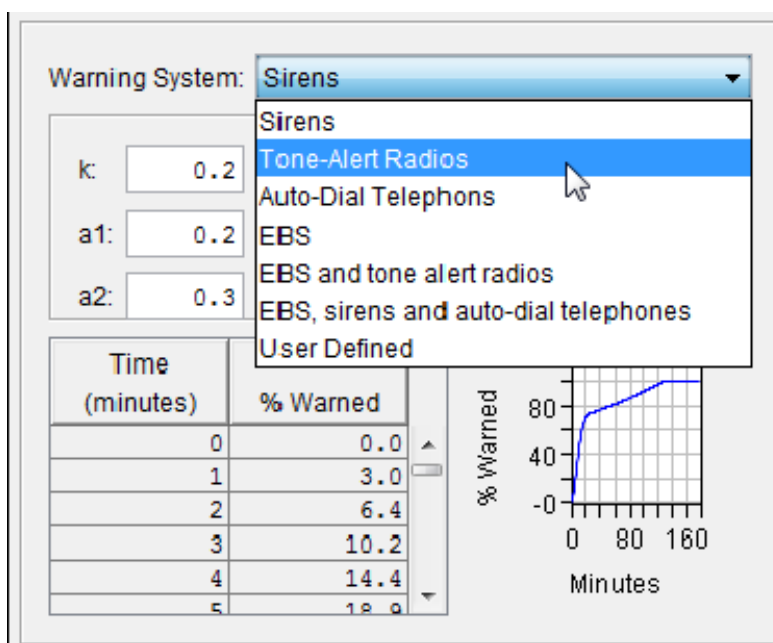
Figure 9.22 Select a Warning System type preset (Impact Area Editor - Life Loss Parameters tab).

5. Once a warning system preset is selected, the function will appear in the table and plotted to the right. Notice that the variable fields will be filled out with the correct values.

6. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, right-click on the thumbnail plot and select **Show Popup** from the dropdown list and the full-size plot will appear in a separate window (Figure 9.23). Alternatively, simply double-clicking on the thumbnail plot will open the full-size plot.

Figure 9.23 View a full size plot (Impact Area Editor - Life Loss Parameters tab).

7. Click **Apply** when finished defining the warning curve. To exit the **Impact Area Editor**, click **OK**.

Users may also create their own warning curve. Five (5) parameters of the warning diffusion equation are user-editable.

To define a custom warning diffusion curve in HEC-FIA:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. From the **Impact Area Name** dropdown list, select the impact area you wish to edit.
3. Select the **Life Loss Parameters** tab**.**

4. From the **Warning System** dropdown list, select the option **User Defined** (Figure 9.24).
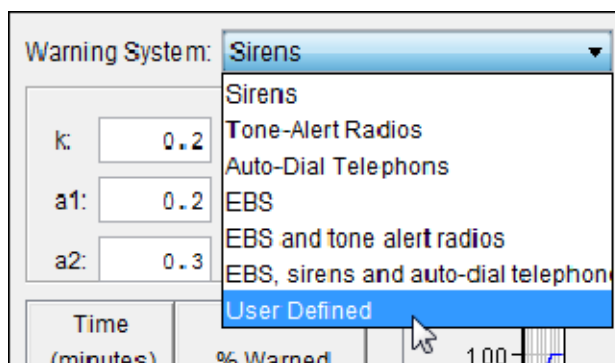
Figure 9.24 Define a custom warning diffusion curve (Impact Area Editor - Life Loss Parameters tab).

5. Using the value entry fields **k**, **a1**, **a2**, **30 Min Limit**, and **Release Rate**, define the various elements of the *Rogers and Sorensen* theorem on warning diffusion (Figure 9.25). For a definition of the theorem and its elements, see Warning Diffusion Concepts, page 9-19. The following explains the functions of the various fields.
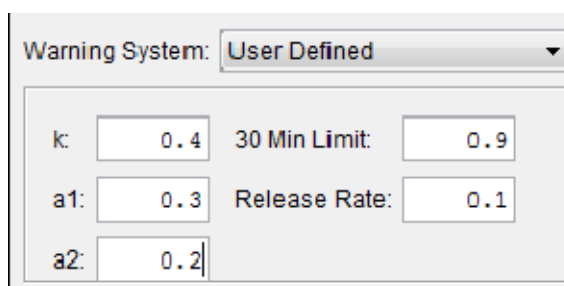
Figure 9.25 Warning function value entry fields (Impact Area Editor—Life Loss Parameters tab).

a. In field **k**, enter the maximum portion of the population that **can** be alerted by and fully understand the warning. The value 1 represents 100 percent of an impact area population. Therefore, the value must be greater than zero and less than or equal to 1. In the example in Figure 9.25, a **k** value of *0.4* has been entered to reflect that *40%* of the population is in a position to be alerted by the warning and fully comprehend its message at the time of your flood event (e.g. if you were defining a warning system such as the Emergency Broadcast system, 40% of the population is watching television).

b. In field **a1**, users define the value of the *Rogers and Sorensen* equation a_1. This value represents the proportion of unwarned individuals warned by the warning system in each time step (t_i to t_{i+1}). For this field, an a_1 value of *0.2* is considered *low*, and a

value of *0.3* is considered *high*. For example, a siren system would be considered low in alerting efficiency (a_1=0.2) because, as a siren cannot convey a direct message, individuals will have to rely on a secondary source to find out why the sirens are sounding and what action is required to avoid danger. But an Auto-Dial Telephone system is considered *very high* in alerting efficiency (a_1=0.35) because the phone message tells individuals what is wrong and what actions are required. In Figure 9.25, a value of *0.3* has been entered to reflect that the user-defined warning system is very efficient in alerting the population to danger.

c.   In field **a2**, users define the value of the *Rogers and Sorensen* equation variable a_2. This value represents the proportion of unwarned individuals warned by secondary means in each time step (t_i to t_{i+1}), i.e. the efficiency of those secondary sources in disseminating the warning message to others. An a_2 value of *0.3* is considered *high*, and a value of *0.2* is considered *low*. For example, the secondary avenues of warning in a siren system would be considered high in efficiency (a_2=0.3) because individuals alerted by the sirens will immediately seek to find out the extent of danger. But an Auto-Dial Telephone system is considered *very high* in alerting efficiency (a_2=0.35) because the phone message itself tells individuals what is wrong and what action is required.

d.   The **30 Minute Limit** defines N, which is the limit of the percentage of population that will be alerted by the warning system and understand its message within 30 minutes of initial warning. For example, in Figure 9.26, a **30 Minute Limit** of 0.75 is selected. At 30 minutes, 75% of the population will have been warned. *1* represents *100%* of the population; therefore, the **30 Minute Limit** should greater than *0* and lesser than or equal to 1.

e.   The **Release Rate** adjusts the overall efficiency of the warning system. With a low release rate (~0.2), the warning diffusion curve will approach the limit of 100 percent very gradually after 30 minutes. With a high release rate (~0.3), the curve will reach the limit of 100 percent over an increasingly short period of time.

6.   As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, right-click on the thumbnail plot and select **Show Popup** from the dropdown list and the full-size plot will appear in a separate window (Figure 9.23). Alternatively, simply double-clicking on the thumbnail plot will evoke the full-size plot.
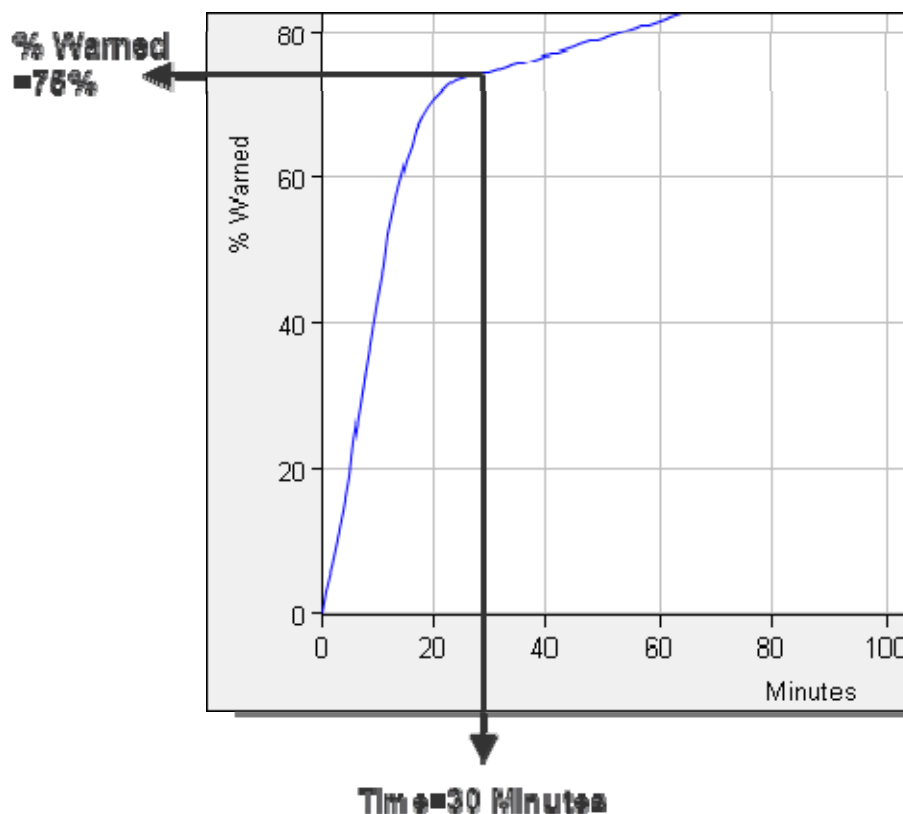
**Figure 9.26 A warning diffusion curve with a 30 Minute Limit (*N* value) of
0.75 will show 75% of the population warned at time=30 minutes
(Impact Area Editor—Life Loss Parameters tab).**

7. Click **Apply** when finished defining warning diffusion data for each
    impact area. To exit the **Impact Area Editor**, click **OK**.

Define a Mobilization Curve

The mobilization curve (Figure 9.27) defines the percentage of population
mobilized over time. Users may choose from one of three presets
(**Default**, **Below Average**, **Above Average**) or define a **User Defined**
curve by editing the values in the table.

To define a mobilization curve:

1. You must be in the **Impact Area Editor**. To access the **Impact
    Area Editor**, see Section 9.7.1 (page 9-8).
2. From the **Impact Area Name** dropdown list, select the impact area
    you wish to edit.
3. Select the **Life Loss Parameters** tab**.**
4. Locate the dropdown list labeled **Mobilization**. From the list, select
    the appropriate mobilization curve preset.
5. Once a mobilization curve preset is selected, the function will
    appear in the table and will be graphed in the thumbnail plot.
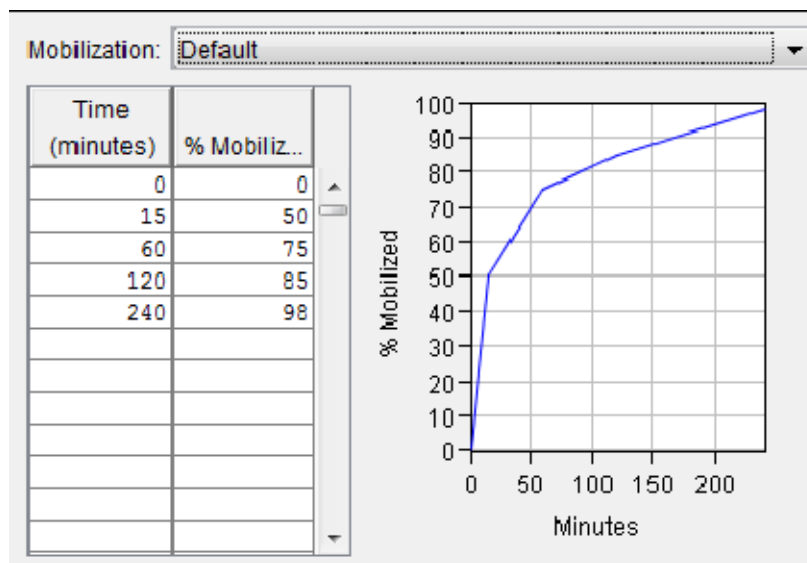
Figure 9.27 Mobilization curve (Impact Area Editor - Life Loss Parameters tab).

6. Alternatively, users may manually define a mobilization function. Under the column labeled **Time (minutes)** enter intervals of time. The first number should be 0 and the last number should be the maximum length of the evacuation time window. In the **% Mobilized** column, enter the number that represents the percentage of population mobilized at each time you defined in the left column. The first value should be 0 and the last value should represent the maximum portion of the population that will be mobilized in a flood event.

7. The thumbnail plot to the right of the table will graph the function defined in the table. To view a full-size plot, right-click on the thumbnail plot and select **Show Popup** from the dropdown list and the full-size plot will appear in a separate window (Figure 9.23, page 9-21). Alternatively, simply double-clicking on the thumbnail plot will evoke the full-size plot.

8. Click **Apply** when finished assigning to impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Safe Zone Fatality Rates

By default, FIA assumes that after a **Non-Evacuation Depth** (typically of 2 feet), most individuals will choose to stay in their structures and face a flood event rather than evacuating. FIA generalizes the location and activities of individuals at different times in a day based on statistical observations. To simulate life loss for those caught in their structures after the flood waters arrive; FIA divides structures into **safe zones** for which users enter different fatality statistics to determine the likelihood that an individual can safely evacuate their structure when flood waters arrive.

Safe zone boundaries are set with default values and modified, if necessary, during the **Structure Inventory** definition process (see Chapter 8, Section 8.11.3). Each safe zone also has default fatality statistics. In the **Impact Area Editor**, users can view and modify **safe zone** fatality statistics as well as activity/location distributions for each **Impact Area**.

To edit fatality statistics for safe zones:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. From the **Impact Area Name** dropdown list, select the impact area you wish to edit.
3. Select the **Life Loss Parameters** tab**.**
4. Click the **Additional Parameters** button at the bottom-right corner of the **Impact Area Editor** (Figure 9.28).
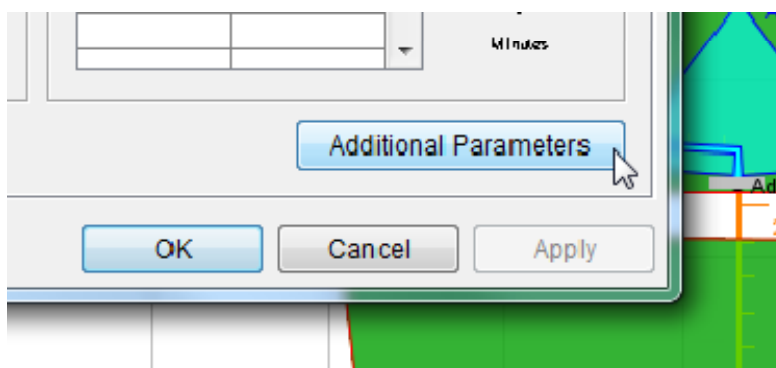
Figure 9.28 Additional Parameters button (Impact Area Editor—Life Loss Parameters tab).

5. The **Additional Parameters** dialog box will open (Figure 9.29).
6. In the **Safe Zone** field, users can modify the fatality rate for the safe zone. The values 0 – 1 represent 0 – 100 % fatality rates for those caught in each zone. The default **Safe Zone** fatality rate is 0. This means that those caught in the **Safe Zone** when flood waters arrive have no chance of fatality. Modify the fatality rate by entering a new value between 0 and 1.
7. In the **Compromised Zone** field, the fatality rate is 0.12. This means that those caught in the **Safe Zone** during a flood event have a 12% chance of fatality. Modify the fatality rate by entering a new value.
8. In the **Chance Zone** field, the default **Compromised Zone** fatality rate is 0.91. This means that those caught in the **Safe Zone** during a flood event have a 91% chance of fatality. Modify the fatality rate by entering a new value.

Figure 9.29 Additional Parameters dialog box.

9. In the field labeled **Non-Evacuation Depth**, enter the inundation depth in feet after which it is assumed that individuals cannot evacuate the structure (Typically, it is assumed that after 2 feet individuals will not risk evacuation.)

10. The **Activity Type Distributions** table shows the statistical distribution of people for 2 hour intervals in a day. The values represent percentages of people occupied in the different activities at that hour of the day (e.g. 91.7% of people are assumed to be at home asleep at 12 AM). Users can modify the distribution by entering a new value between 0 and 100.

11. Once all desired changes have been made, click **OK**. The **Additional Parameters** dialog box will close and you will be back in the **Life Loss Parameters** tab of the **Impact Area Editor**.

12. Click **Apply** when finished assigning safe zone fatality rates to impact areas. To exit the **Impact Area Editor**, click **OK**.

9.7.7   Damage Functions Tab

In the **Damage Functions** tab (Figure 9.30), users enter elevation relationships for area inundated, urban damage, number of structures inundated, and population displaced for a generalized structure inventory (not added using gridded data). For number of structures inundated and urban damage, users can enter elevation functions either by damage categories or by aggregated total damage.

This function is only pertinent to **Time-Series-Based Alternatives** (concepts in Section 9.7, page 9-7).

*The **Damage Functions** tab is where users identify the elevation relationships for area inundated, number of structures inundated, population displaced, and urban damage for an impact area.*
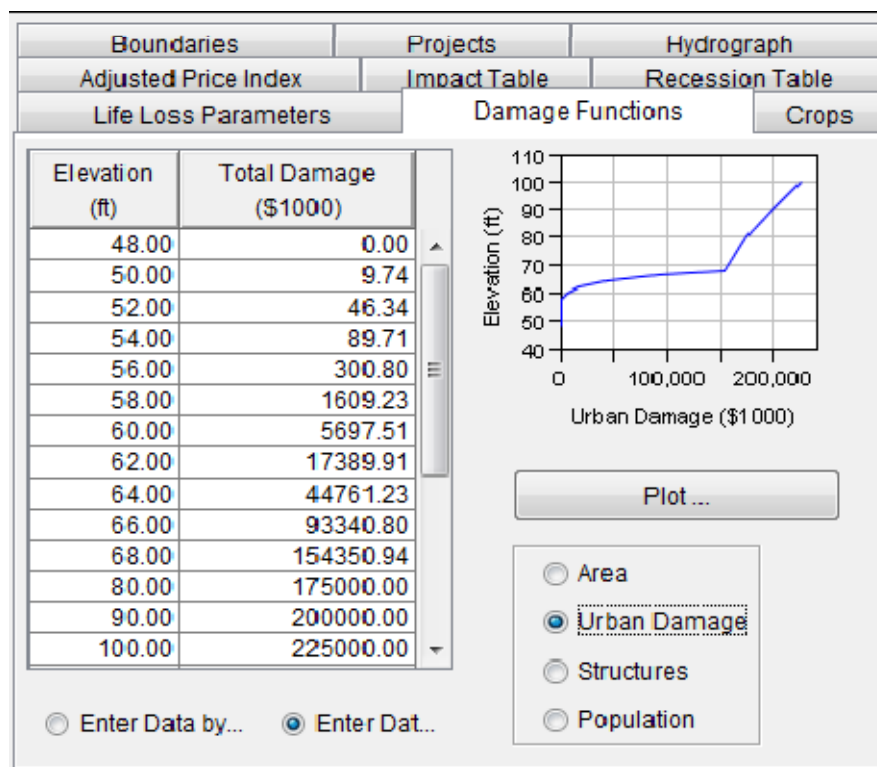
Figure 9.30 Impact Area Editor—Damage Functions tab.

Definition of data entry items follows:

*An **elevation-area function** describes the topography of the impact area, relating elevation to the area that could be inundated. FIA uses this function for two calculations: calculating the area inundated and for computing agricultural damages.*

- **Area** – elevation-area function, relates water-surface elevation in an impact area to the area of land, in acres (hectares), that is inundated. FIA uses this relationship when computing the area of a specific crop that is flooded.
- **Elevation** – a range from the lowest to highest point in an impact area. These values are also used to define the flood zones for the damage computations for different duration's of flooding. In FIA, when the set of elevations is defined, the set will be used for all of the damage relationships for an impact area.
- **Population** – elevation-population function, relates the water-surface elevation in an impact area to the population displaced in that impact area.
- **Structures** - elevation-structure function, relates the water-surface elevation in an impact area to the number of structures inundated in that impact area.
- **Urban Damage** – elevation-damage function, relates the water-surface elevation in an impact area to the structure, content, and infrastructure damage incurred in that impact area.

Define Elevation-Area Function

To define an **elevation-area function**:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab.
3. Click **Area**. The table will have two columns with the headings **Elevation** and **Area** (Figure 9.31).
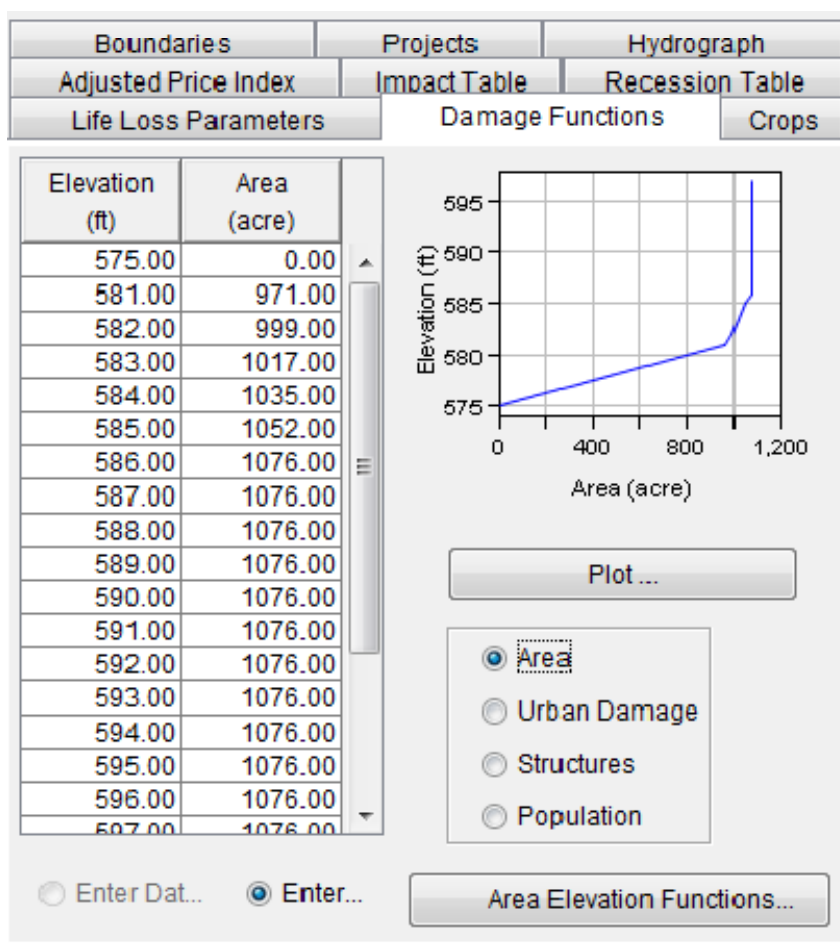
Figure 9.31 Elevation-area function (Impact Area Editor—Damage Functions tab).

4. Enter values that will create the **elevation-area function** or paste them from an external spreadsheet (Figure 9.31). The tab will expand to fit pasted objects. The numbers typed in the **Elevation** column must be referenced to the same datum as your stage hydrograph and/or rating curve for the active impact area. In the **Area** column, the first number should be zero, and the last number should correspond to the area of the entire impact area.
5. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot**.
6. Click **Apply** when finished defining elevation-area functions for impact areas. To exit the **Impact Area Editor**, click **OK**.

Generate Elevation-Area Function from Shapefile

In HEC-FIA 2.1, users also have the option of generating an Elevation-Area function from a shapefile with crop information. This option will automatically calculate Elevation-Area values based on a crop shapefile and a terrain grid.

To generate elevation-area functions from shapefile:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab.
3. To edit the elevation-area damage function, select **Area** from the damage function options (Figure 9.32).
4. Beneath the damage function options, select the **Area Elevation Functions...** button (Figure 9.32).
5. The **Generate Area Elevation Functions from Crop Shapefile** dialog box will appear.
6. From the dropdown list labeled **Crop Shapefile**, select the shapefile you wish to associate with crop data (required).

7. From the **Terrain Grid** dropdown list, select the SMRMP file to use as a terrain model (required).

Figure 9.32 Generate elevation-area functions from shapefile (Impact Area Editor—Damage Functions tab).

8. From the **Impact Area** dropdown list, select the Impact Area for which an elevation-area function is to be generated (required).
9. Using the **Elevation Resolution** box, users may set the grid cell resolution for each crop polygon. If left at 0, HEC-FIA will evaluate each crop polygon based on the terrain grid cell resolution to calculate an elevation-area function. The value in the **Elevation resolution** box sets a grid cell resolution within each crop polygon. For example, if the user enters an elevation resolution of 5, HEC-FIA would divide each crop shapefile into 5 by 5 grids for evaluation. Each individual 5 by 5 grid cell then samples the elevation grid and assigns that elevation to the area represented by the 5 by 5 grid cell. This is designed so that users can control the level of detail in their study.
10. Under the **Data Assignments** section, select the crop for which you want to calculate damages by area (required).
11. To apply this elevation-area to all impact areas within your study, select the option **Apply to all Impact Areas**. If left unchecked, the

*An **elevation-damage function** describes the relationship of urban economic damage (inundation damage, emergency costs, etc.) to elevation in an impact area. Damage represented by these functions can be an aggregated (total) or by damage category.*

elevation-area function from shapefile will apply only to the impact area you selected in step 8.

12. Once all data has been entered, click **OK.** The elevation-area function generated from shapefile will now appear in the damage function tab of the **Impact Area Editor**.

Define Elevation-Damage Function

To define an **elevation-damage function**:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab.
3. You can have an elevation-damage function for each damage category (**Enter Data by Damage Category**) or you can enter one aggregated elevation-damage function (**Enter Data by Total Damage**) for the active impact area.
4. Select **Urban Damage**. For the example in Figure 9.33, an aggregated elevation-damage function is entered. Click **Enter Data by Total Damage** (the radio button on the bottom-right). The table will have two columns with the headings **Elevation** and **Total Damage**.
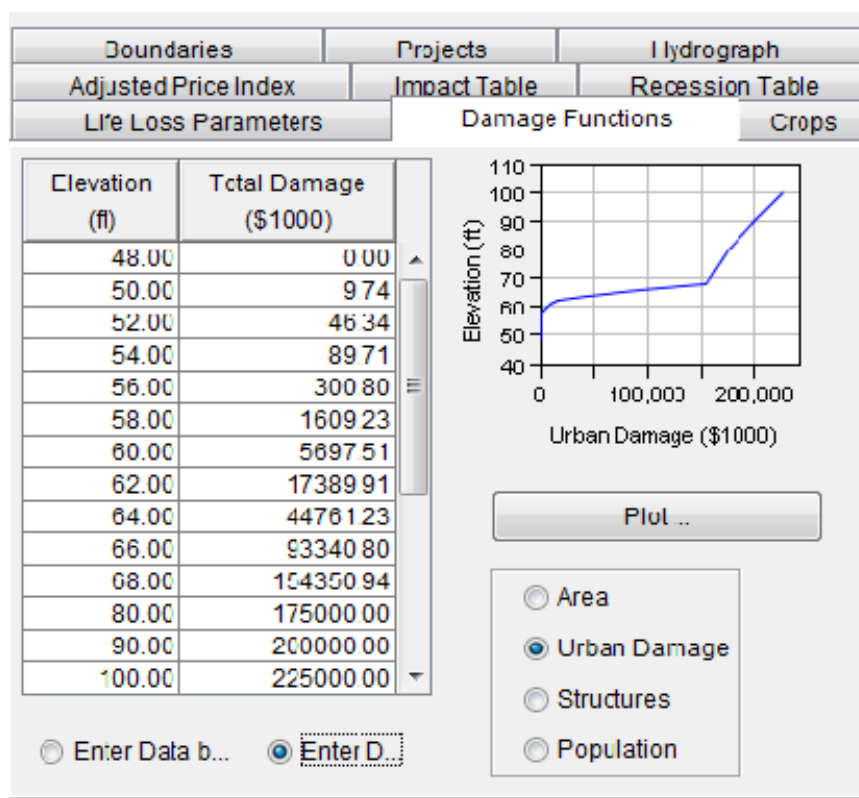
Figure 9.33 Elevation-damage function (Impact Area Editor - Damage Functions tab).

5. If you have already defined an elevation-area function, the **Elevation** column will already have values entered. Then you will only need to enter values in the **Total Damage** column. FIA remembers the elevation numbers that have been entered.
6. Enter the values that will create the **elevation-damage function** or paste them from an external source (Figure 9.33). In the **Total Damage** column, the first number should be zero, and the last number should be the maximum damage in the impact area.
7. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot**.
8. Click **Apply** when finished defining elevation-damage functions for impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Elevation-Structure Function

*An **elevation-structure function** identifies the number of structures that could be inundated as water reaches specified elevations. As with the urban damage functions, the impacts represented can be aggregated (total) or by damage category.*

To define an **elevation-structure function**:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab. You can have an elevation-structure function for each damage category created in the **Global Data Editor** (**Enter Data by Damage Category**) or you can enter one aggregated elevation-structure function (**Enter Data by Total Damage**) for the active impact area.
3. Click **Structures**. For the example in Figure 9.34, you will be entering an aggregated elevation-structure function.
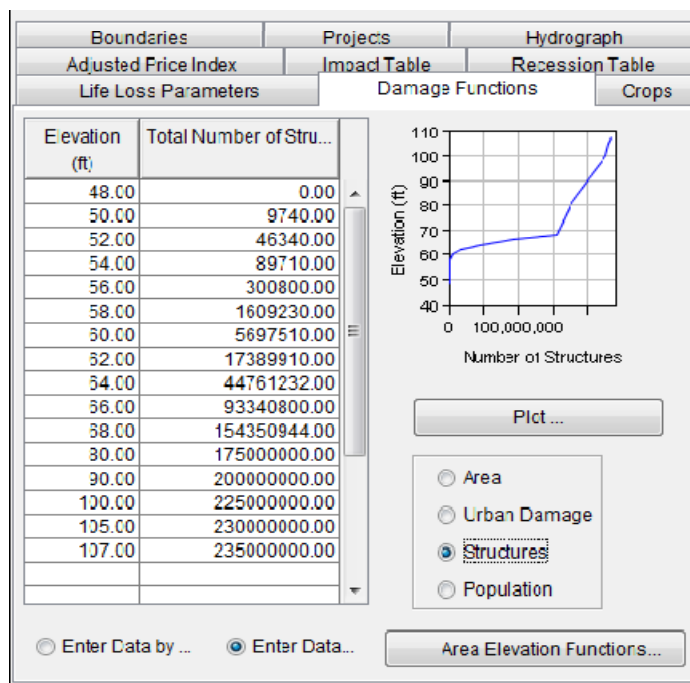
Figure 9.34 Elevation-structure function (Impact Area Editor - Damage Functions tab).

4. If you have already defined an elevation-area function, the **Elevation** column will already have values entered. Then you will only have to enter values in the **Total Number of Structures Impacted** column to create the elevation-structure function. FIA remembers the elevation numbers that have been entered.
5. Enter the values that will create the **elevation-structure function** or paste them from an external source (Figure 9.34, page 9-32). In the **Total Number of Structures Impacted** column, the first number should be zero, and the last number should include all structures potentially at risk in the impact area.
6. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot**.
7. Click **Apply** when defining elevation-structure functions for impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Elevation-Population Function

To define an **elevation-population function**:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab.
3. Click **Population**. The table should have two columns with the headings **Elevation** and **Total Population Impacted**.
4. If you have already defined an elevation-area function, the **Elevation** column will already have values entered. Then you will only need to enter values in the **Total Population Impacted** column. FIA remembers the elevation numbers that have been entered.
5. Enter the values that will create the **Elevation-Damage Function** or paste them from an external source. In the **Total Population Impacted** column, the first number should be zero, and the last number should include all population potentially at risk in the impact area.
6. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot**.
7. Click **Apply** when defining elevation-population functions for impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Damage Function by Damage Categories

There are two options for damage function definition in the **Damage Functions Tab** of the **Impact Area Editor** available at the bottom of the dialog box. If users select **Enter Data by Total Damage**, the function will measure the aggregated structure damages (i.e. the damages to all structures) at each elevation. Alternatively, if users select the option **Enter Data by Damage Category**, they can define separate aggregated

elevation-damage functions for each of the **Damage Categories** they created in the **Global Data Editor** (Chapter 4, Section 4.4). To define any damage function by **damage categories**

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Damage Functions** tab.
3. Select the damage function you wish to define (i.e. *Area, Urban Damage, Structures, Population*). In the example, *Urban Damage* has been selected to define an **Elevation-Damage Function**.
4. Click **Enter Data by Damage Category**. The table will have an **Elevation** column and a column for each **Damage Category** defined in the **Global Data Editor**. In the example in Figure 9.35, four columns for damage categories are shown.
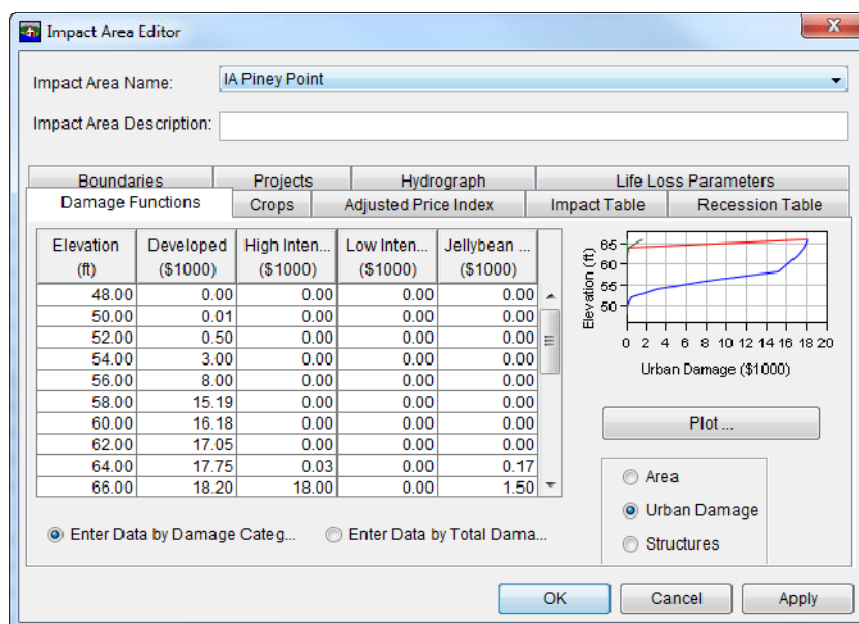
Figure 9.35 Elevation-damage function by damage categories (Impact Area Editor - Damage Functions tab).

5. If you have already defined an elevation-area function, the **Elevation** column will already have values entered. Then you will only need to enter values in the individual damage category columns. FIA remembers the elevation numbers that have been entered.
6. Enter the values that will create the damage function you have selected or paste them from an external source (Figure 9.35). In each damage category column, the first number should be zero, and the last number should be the maximum damage in the impact area.
7. As values are entered, the thumbnail plot to the right of the table will continually update. To view a full-size plot, click **Plot**.
8. Click **Apply** when finished defining damage functions for impact areas. To exit the **Impact Area Editor**, click **OK**.

9.7.8   Crops Tab

In the **Crops** tab (Figure 9.36), users define the crop patterns for the impact area. The crop patterns define the spatial distribution of crops throughout the range of elevations in the impact area. When a simulation is computed, the elevation-area relationships will be used to determine the cultivated area flooded. Along with the crop information you entered in the **Global Data Editor**, FIA can use these area values to compute agricultural flood damage.

Figure 9.36 Impact Area Editor—Crops tab.

This function is only pertinent to **Time-Series-Based Alternatives** (concepts in Section 9.7, page 9-7).

There are two possible types of crop patterns – **Common** and **Unique**. Definition of data entry items follows:

- **Common** – crop pattern, crop distribution is constant over the entire agricultural area.
- **Crops** – crop names defined in the **Global Data Editor** (Chapter 4, Section 4.5).
- **Associated With** – check the boxes for each crops planted in the selected impact area.
- **Percent of Total Area** – the percentage of total area that a particular crop covers in an impact area. A value of one hundred percent represents the entire impact area. The individual crop percentages do not need to add up to one hundred, but the percentages cannot exceed one hundred percent. This method does

not differentiate as to whether the crop is growing in the upper or lower area of the impact area.

● **Unique** – crop pattern, crop varies over the agricultural area elevation range. Use this crop pattern where a uniform distribution is not representative of the crop distribution pattern for the impact area. For example, lower areas that flood more frequently might be planted in a more water tolerant crop, whereas adjacent higher grounds are planted in another crop.

● **Use Double Cropping** – use this option when a second crop can be planted in the same field during the same year after the harvest of the first crop.

Define Uniform Crop Distribution

To define a uniform crop distribution over an entire impact area:

*The **Common** option is a crop pattern where the crop distribution is constant over the entire agricultural area.*

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Crops** tab.
3. Click **Common** (default). The table (see Figure 9.36) contains a list of crops you will select from to identify those that are planted within that particular impact area.
4. To select a crop, check the box in the **Associated With** column of the table.
5. In the **Percent of Total Area** column, type a number that represents the percentage of the total area that the particular crop covers in the impact area.
6. Repeat steps 4 and 5 for each crop you wish to associate with your impact area.
7. Click **Apply** when finished assigning to impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Variable Crop Distribution

To define a crop distribution that varies over the agricultural area's elevation range:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Crops** tab.
3. Click the **Unique** option (Figure 9.37). From the available crops, define the unique elevation-area functions for each crop within the impact area.
4. You have already defined an elevation-area function in the **Damage Functions** tab. Type the values that will create an **Elevation-Area Function** (Figure 9.37) for each crop within the impact area.

Figure 9.37 Define variable crop distribution (Impact Area Editor - Crops tab).

*The **Unique** option is a crop pattern in which the crop distribution varies over the agricultural area elevation range. Use this crop pattern where a uniform distribution is not representative of the crop distribution pattern for the impact area.*

5. Click **Apply** when finished assigning to impact areas. To exit the **Impact Area Editor**, click **OK**.

Define Crop Distribution with Double-Cropping

Double cropping is when a second crop is planted in the same field during the year after the harvest of the first crop.

To define double cropping over the entire area:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Crops** tab.
3. Check the box labeled **Use Double Cropping**, and then click **Edit.** The **Edit Double Crops** dialog box (Figure 9.38) will open.
4. Crops entered in the **Global Data Editor** (Chapter 4, Section 4.5.3), can be designated primary or secondary. If there is double cropping for the selected impact area, you must associate the secondary crops with the primary crops.

Figure 9.38 Edit Double Crops dialog box.

5. In Figure 9.38, *Wheat* is a primary crop, with the secondary crop being *Spring Wheat*.

6. Click **OK**, the **Edit Double Crops** dialog box will close. The **Common** and **Unique** crop patterns will now list the primary crops followed by their secondary crop. For example, in Figure 9.39, the row is now labeled *Wheat/Spring Wheat*.

Figure 9.39 Common Crop pattern using double-cropping (Impact Area Editor—Crops tab).

9.7.9   Adjusted Price Index Tab

The **Adjusted Price Index** tab contains the same index factor information entered in the **Global Specs.** section of the **Global Data Editor** (Chapter 4, Section 4.2, page 4-2). Use this tab only if you want to override the values entered on the **Global Specs.** tab for a particular impact area.

To override the global index information for an impact area:

> *The Adjusted Price Index tab is where users adjust the base data year and the price index factor for both urban and agricultural specifications by impact area. These values will override the global base data year and price index factor.*

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).

2. Click **Adjusted Price Index** tab.

3. For example, say the **Urban Specifications Price Index Factor** value was set in the **Global Data Editor** as *1.17*. In Figure 9.40, the **Price Index Factor** is changed to *1.25*.

Figure 9.40 Price Index Factor change (Impact Area Editor—Adjusted Price Index tab).

4. Click **Apply** when finished editing data. To exit the **Impact Area Editor**, click **OK**.

9.7.10   Impact Table Tab

The **Impact Table** tab (Figure 9.41) contains a flood-preparedness plan for an impact area. This table is optional and can be used to define a set of rules for flood-threat recognition, warning dissemination, emergency response, and post-flood recovery and re-occupation of the impact area.

*The **Impact Table** tab is where you can define a set of rules for flood-threat recognition, warning dissemination, emergency response, and post-flood recovery and re-occupation of an impact area.*

Figure 9.41 Impact Area Editor—Impact Table tab.

Each row of the table specifies one rule, in *if...then...* format. Column 1 specifies the *if...* part of the rule, i.e. the stage that constitutes some level of flood threat for the impact area shown. The stage values do not have to be the same as those entered for the damage/impact functions. Column 3 specifies the *then...* component of the rule; the action that should be taken to respond to that particular threat. Column 2 of the table identifies the problem relating to the given stage.

To enter the impact response table for an impact area:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click the **Impact Table** tab.
3. In Figure 9.41, an impact response table has been entered for the *IA Piney Point* impact area. The second row represents the following rule:

> **If** *the observed stage at the Lock Haven index station exceeds 70 feet* **then** *the Public Affairs Office will order a mandatory evacuation, as moderate urban flooding has begun.*

As noted earlier in this manual, FIA can simulate conditions and evaluate damage in an impact area during a flood event using time-series hydrographs. As the program steps through this simulation, evaluating damage, it also searches through the rules in the impact table to identify the stage thresholds that are exceeded. When it finds a threshold that is

exceeded, it records the time and date. Then, at the conclusion of the simulation, FIA will create an **Impact Response Report** (see Chapter 12, Section 12.8, Page 12-21). These rules will not affect damage computation; FIA will simply report the time and date each action you enter here should be accomplished.

9.7.11   Recession Table Tab

The **Recession Table** tab (Figure 9.42) is a preparedness plan for an impact area that gives alerts potential environmental and operational problems due to low flow. This table is optional and can be used to define a set of rules for warning dissemination, and environmental and infrastructure response.

*The **Recession Table** tab is where you can define a set of rules for warning dissemination, and environmental and infrastructure response of an impact area.*

Stage (ft)	Stage Impact	Action Required
575	Flood Stage	Minor agricultural flooding begins
550	Not enough flow to maintain irrigation supplies	Turn off irrigation pumps
541.5	Poor oxygen content, possible fish kill	Oxygen infusion
515.0	Not enough flow to maintain municpal water supply	Turn off pumps; Increase
500	Endangered species threatened	Increase water supply

Figure 9.42 Impact Area Editor - Recession Table tab.

Just as in the **Impact Table** (Section 9.7.10, page 9-39), each row of the table specifies one rule, in *if...then...* format. Column 1 specifies the *if...* part of the rule, i.e. the stage that constitutes some level of low flow for the impact area shown. The stage values do not have to be the same as those entered for the damage/impact functions. Column 3 specifies the *then...* component of the rule; the action that should be taken to respond to that particular threat. Column 2 of the table identifies the problem relating to the given stage.

To enter the recession response table for an impact area:

1. You must be in the **Impact Area Editor**. To access the **Impact Area Editor**, see Section 9.7.1 (page 9-8).
2. Click **Recession Table** tab.
3. In Figure 9.42, a recession response table has been entered for the *Beech CK Agri* impact area. The second row represents the following rule:

    *If the observed stage at the Beech CK Agri index station meets 550.00 feet **then** turn off the irrigation pumps, as there is not enough flow to maintain irrigation supplies.*

As noted earlier in this manual, FIA can simulate conditions and evaluate damage in an impact area during a flood event using time-series hydrographs. As the program steps through this simulation, evaluating

damage, it also searches through the rules in the recession table to identify the stage thresholds need to be maintained or exceeded. When it finds a threshold that is not met, it records the time and date. Then, at the conclusion of the simulation, FIA will create a **Receding Response Report** (Chapter 12, Section 12.9, page 12-22).

9.8   Edit HEC-DSS Mappings

If users are employing a time-series-based alternative, once they have created the alternative and entered hydrograph information (Section 9.7.4) and damage function data (Section 9.7.7), they will need to link time-series data to the alternative.

Additionally, if users are entering project benefits using the option **Specify Regulated and Unregulated Time-Series** (see Project Benefits, page 9-12), they will need to manually allocate reservoir benefits to each reservoir.

9.8.1   Link Time-series Data

After an alternative has been created, you can now link time-series data to the hydrograph locations (computation points) that were identified in the **Impact Area Editor**. You will be using HEC-DSSVue to link your time-series data.

To link time-series data:

1. Starting in the **Study Pane**, right-click on the alternative you wish to edit. From the shortcut menu (see **Error! Reference source not found.**, page 9-6), select **Edit HEC-DSS Map**.
2. The **Edit Time-series Mappings** window will appear, displaying computation point locations and associated DSS file information in your study (Figure 9.43).
3. Click on a row and in the bottom right corner of the **Edit Time-series Mappings** dialog box, select **Browse DSS Paths**. The **Select Time-series Path** dialog box will open (Figure 9.44).
3. Open the HEC-DSS file that contains the flow or stage time-series data for the watershed by clicking the open folder icon 🖻 and using the **Open File** browser to navigate to and select the proper HEC-DSS file. In Figure 9.44, a DSS file has been selected. All the pathnames from the selected HEC-DSS file are now shown in the **Pathname Table** of the **Select Time-series Path** dialog box.
4. The use of filters makes it easier for you to locate the appropriate pathname. An example of using filters would be for **Filter B**, from the list select *IA_Barker*, for **Filter C**, from the list select *ELEVATION-DAMAGE*, and for **Filter F**, from the list click *DAMAGE FUNCTIONS*.
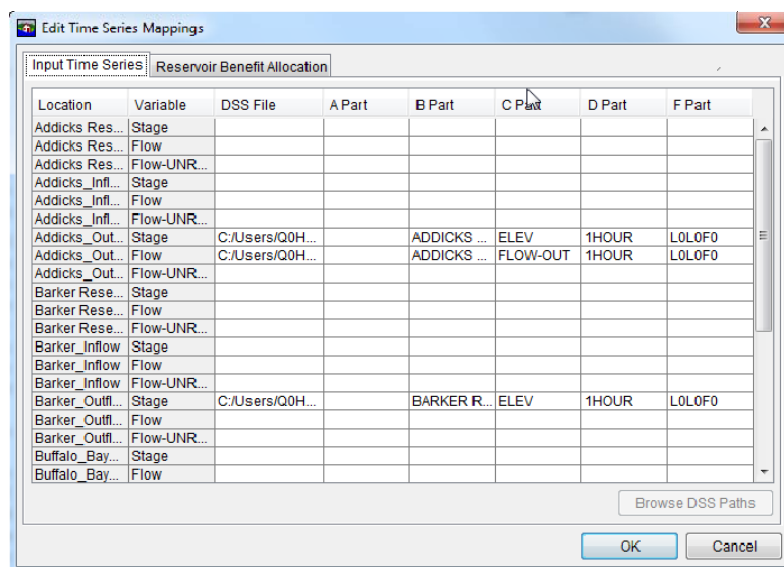
Figure 9.43 Edit Time-series Mappings dialog box.
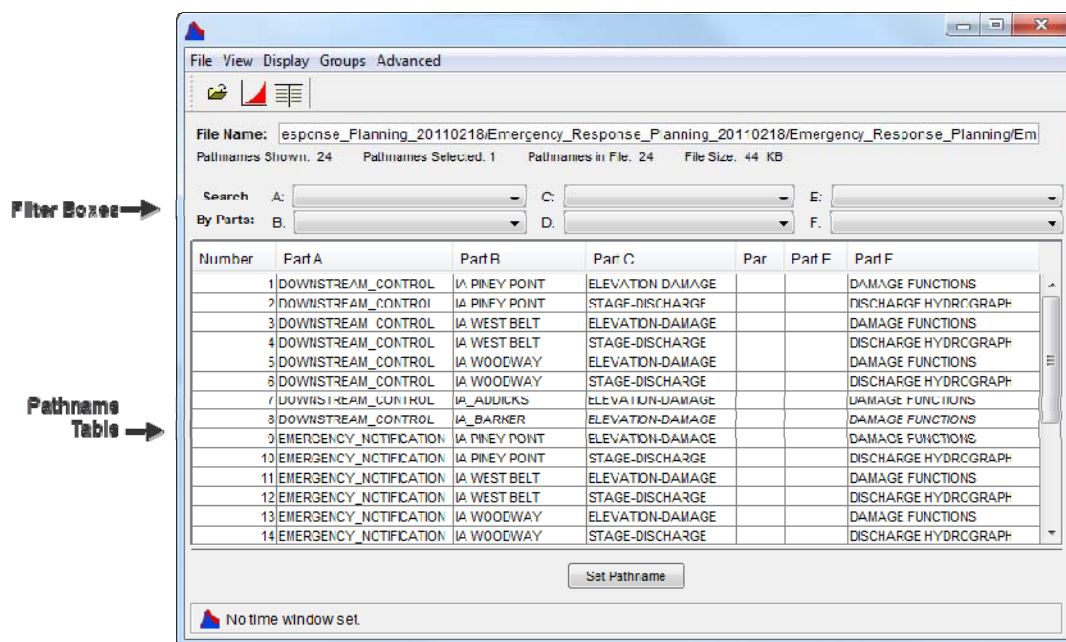
Figure 9.44 Select Time-series Path dialog box.

5. Click on the appropriate pathname, the row will be highlighted in blue.

6. Click **Set Pathname**. This will copy the selected pathname into the row you selected in step two in the **Edit Time-series Mappings** window. Double-clicking on the DSS pathname will also assign it to a row in the **Time-series Table**.

7. In the **Edit Time-series Mappings** window, click on the next row to which you want to assign a pathname. The **Select Time-series Path** dialog box will remain open, click on it to make it active. Repeat steps 5-6 for each row.

8. When you are done, from the **Select Time-series Path** dialog box, on the **File** menu, click **Close**. This will close the **Select Time-series Path** dialog box.

9.8.2   Reservoir Benefit Allocation

If project benefits are to be computed for an alternative using the option **Specify Regulated and Unregulated Time-Series – Assign Reservoir Benefits (Define for each run)** (see Project Benefits, page 9-12), users will need to manually allocate reservoir benefits to each reservoir. Users should only allocate project benefits as percentages if they have verifiable data outside of their FIA study that demonstrates project benefits. Generally it is best to let FIA compute project benefits (see Appendix C for details on project benefit calculation). This action is also performed from **Edit Time-series Mappings** dialog box (Figure 9.43).

1. Starting in the **Edit Time-series Mappings** dialog box, select the **Reservoir Benefit Allocation** tab.
2. This tab (Figure 9.43) contains a table with the impact areas for which a user has assigned project benefits using the option **Specify Regulated and Unregulated Time-Series** (see Project Benefits, page 9-12) along with the reservoirs in the study.

3. Under the % Allocation column, users must enter the percentage of benefits (the percentage of damage prevented by the project). In Figure 9.45, *50* has been entered for *Addicks Reservoir* in the *IA Piney Point* impact area to reflect that the Addicks Reservoir will reduce damage to this impact area by 50% from the damage that would occur without any projects.
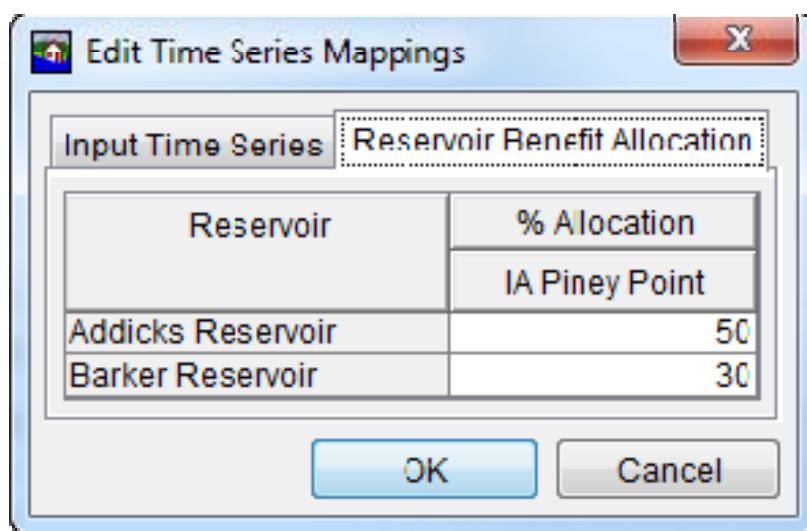
Figure 9.45 Edit Time-series Mappings dialog box - Reservoir Benefit Allocation tab.

4. Repeat steps 1 – 3 for each alternative in your study that has been flagged for project benefits using the **Specify Regulated and Unregulated Time-Series** option in the **Impact Area Editor**.

Chapter 10

Time Window

Once you have created alternatives, you can begin to create **Simulations** to model flood events and estimate consequences. But before you create a Simulation, you must define a **Time Window** that a simulation can operate within.

10.1   Concepts

A **Time Window** defines the time that a simulation will start and finish. With the **Time Step** option, it also defines the level of detail a simulation will model flood events. A simulation using a **Time Step** of one hour will model events and consequences at every hour within the **Time Window**; a simulation using a **Time Step** of fifteen minutes will analyze the same data every fifteen minutes within the same window, creating a more rigorous simulation.

A **Time Window** is also required for **Life-Loss Computation**. In Chapter 9, Section 9.7.5, you learned how to define the life-loss parameters of the **Impact Area Editor** to be used in life-loss consequence assessment. To simulate life-loss, users must create a **Time Window** that defines the date and time the issuance of a warning—the initial time (t_0) of warning diffusion—will begin. Users can define a warning issuance date and time for an entire structure inventory or specify different warning issuance dates and times for each **Impact Area**, **Warning Region**, or even by **Individual Structures**. Or users can create multiple **Time Windows** to simulate life loss by all of these different units.

Definition of data entry items are as follows:

- **Start Date/ Time**: (required) this is the date and time on which FIA will begin computing damage for a simulation.
- **Lookback Date/ Time:** (optional) this is the date and time on which FIA will begin tracking the damage potential for the watershed. FIA will not begin computing damages until the start date and time. If a flood occurred prior to the start date that affects the damage potential on the start date, (for example a crop area is still in the dryout period), FIA will take that flood into account through the lookback date and time. If you do not specify a lookback data and time, FIA will assume it is the same as the start date and time.

- **End Date/Time:** (required) this is the date and time on which FIA will stop computing damage for the simulation.
- **Time Step:** this is the time interval or time step of the time series data in a DSS file. The time intervals available are: *15 minutes, 30 minutes, 1 hour, 3 hours, 6 hours, 12 hours*, and *1 day*. The default is *1 hour*. This is required for a time-series based compute but is inconsequential to a grid-based compute.

The following data entry fields are required for Life Loss Compute only, and thus unnecessary for a compute without life loss information:

- **Breach Initiation Date/Time:** (for information only) this is the date and time that a flood event has occurred; where, for example, a levy was overtopped or a reservoir was breached. It is not required for life loss compute, but can be entered for completeness
- **Define Time By:** (required for Life Loss Compute) this option allows users to define different **Warning Issuance Dates/Times** by a variety of different units: **All Structures**, meaning the entire structure inventory; **Individual Structure**, setting a different **Warning Issuance Date/Time** for each individual structure in an inventory; **Impact Area**, by each impact area; and **Warning Region**, using warning regions derived from shapefiles
- **Warning Issuance Date/Time:** (required for Life Loss Compute) this sets the initial date and time a warning has been issued to the public, and thus is the date/time that FIA will begin simulating life loss consequence

10.2   Create Time Window (no Life-Loss Simulation)

To create a **Time Window** without Life-Loss simulation

1. Starting in the **Study Pane**, locate the **Time Window** folder and right-click. From the shortcut menu click **New** (Figure 10.1).

Figure 10.1 Create new Time window (Study Pane).

2. The **Create New Time Window** dialog box will open (Figure 10.2). In the **Name** box enter a name for the time window.
3. In the **Description** box enter a short description.
4. In the **Directory** box, either enter or browse to the directory where you would like to store the time window. It is advised to accept the default (i.e., *C:\Users\Example\Documents\FIA\FRM_Example*) which is the study directory.
5. For the start date you will enter a date using the format *ddmmmyyyy* (i.e. 23Mar2011) in the **Start Date** box (Figure 10.2). The start time

Figure 10.2 Create New Time Window dialog box, no life loss information entered.

is entered in twenty-four hour clock time (i.e. 2200) in the **Time** box.

6. For the lookback date (optional), enter a date using the format *ddmmmyyyy* in the **Lookback Date** box. The lookback date must be less than or equal to the start date. The lookback time is entered in twenty-four hour clock time in the **Time** box. See Section 10.1, page 10-1 for an explanation of lookback dates.

7. For the end date, enter a date using the format *ddmmmyyyy* in the **End Date** box. The end date must be greater than the start date. The end time is entered in twenty-four hour clock time in the **Time** box.

8. The **Time Step** (Figure 10.2) is not used for a grid-based compute and should be left at the default value (*1 Hour*). If performing a time-series-based compute, select a **Time Step** for interpolation of the DSS data.

9. As this **Time Window** is not using life-loss calculation, leave the fields designated **Required For Life Loss Compute Only** blank. See the next section for creating a **Time Window** with life-loss calculation.

10. Click **OK** (Figure 10.2). The **Create New Time Window** dialog box will close. The time window has been created and the name will appear in the **Study Pane** under the **Time Window** folder.

10.3   Create Time Window with Life Loss Simulation

You have defined life loss information in the **Impact Area Editor** (see Chapter 6, Section 6.9.5. To simulate life loss, you must create a **Time Window** for that information to operate within. The **Time Window** parameters pertaining only to life loss computation are located at the bottom of the **Create New Time Window** dialog box (see Figure 10.3).

Figure 10.3 Time window to compute life loss information (Create New Time Window dialog box).

To create a **Time Window** with life loss simulation:

1. Starting in the **Study Pane**, locate the **Time Window** folder and right-click. From the shortcut menu click **New** (see Figure 10.1, page 10-2).
2. Complete steps 2 through 8 in the preceding **Section 10.2**, starting on 10-2.
3. In the **Breach Initiation Date** box click the ⊡ browse button to select the calendar date on which the flood began or enter the date in *ddmmmyyyy* format. In the **Time** box enter the 24 hour clock time the flood event initiated (see Figure 10.3).
4. From the **Define Time By** dropdown list, select one of the categories for which you would like to define a **Warning Issuance Date/Time**. There are four mutually exclusive options: **All Structures**, **Individual Structure**, **Impact Area**, and **Warning Region**. The process for defining a date/time of warning issuance for each of these options is explained in the following points (a) through (d)
   a. *Define Time By→ All Structures*. Select this option to define a *single* warning issuance time for *your entire study.* In the **Warning Issuance Time** box, click the ⊡ browse button to select the calendar date an evacuation warning is to be issued or enter the date in *ddmmmyyyy*

format. In the box to the right enter the 24 hour clock time the evacuation warning will be issued (see Figure 10.3).

b. *Define Time By*➤*Individual Structure*. Select this option to define *separate* warning issuance times for *each structure in your structure inventory* (see Chapter 8 for information on structure inventories)*.* When you select the option **Define Time By Individual Structure,** a three column table will appear (Figure 10.4).

Figure 10.4 Define Time By Individual Structure (Create New Time Window dialog box).

In the column labeled Structure Name is a list of every structure in your Structure Inventory. Users may enter a different Warning Issuance Date/Time for each of these structures if desired. In the W**arning Issuance Date column**, click on a cell to select the calendar date an evacuation warning is to be issued or enter the date in ddmmmyyyy format. Under the **Warning Issuance Time** column enter the 24 hour clock time the evacuation warning will be issued (Figure 10.4).

c. *Define Time By*➤*Impact Area*. Select this option to define *separate* warning issuance times for *each Impact Area* in your study*.* When you select the option **Define Time By Impact Area,** a three column table will appear (Figure 10.5).

In the column labeled Impact Area is a list of the Impact Areas in your study. Users may enter a different Warning Issuance Date/Time for each of these Impact Areas if desired. In the Warning Issuance Date column, click on a

Figure 10.5 Define Time By Impact Area (Create New Time Window dialog box).

cell to select the calendar date an evacuation warning is to be issued or enter the date in ddmmmyyyy format. Under the Warning Issuance Time column enter the 24 hour clock time the evacuation warning will be issued (Figure 10.5).

d. *Define Time By➔Warning Region.* Select this option to define separate warning issuance times for each **Warning Region** in your study. When you select the option **Define Time By Impact Area**, two dropdown lists will appear in addition a three column table (Figure 10.6).

Figure 10.6 Define Time By Warning Region (Create New Time Window dialog box).

In the field labeled **Warning Region**, select the shapefile that contains warning region information (e.g *warning_regions.shp*). From the **Attribute** dropdown list, select the data entry field of the shapefile to which you

would like to assign **Warning Issuance Dates/Times**. These will appear in the **Warning Region** column of the three-column table. In the example in Figure 10.6, there is only one shapefile attribute (*Name*) and it has been selected so that the names of each warning region (*Floodplain, Sauvie Island,* and *Woodland*) are now in the leftmost column of the table.

In the column labeled **Warning Region** is the shapefile attribute you selected in the previous step**.** Users may enter a different **Warning Issuance Date/Time** for each of these **Attributes** if desired. In the **Warning Issuance Date** column, click on a cell to select the calendar date an evacuation warning is to be issued or enter the date in *ddmmmyyyy* format. Under the **Warning Issuance Time** column enter the 24 hour clock time the evacuation warning will be issued (Figure 10.6).

5. Now that you've selected one of the warning time definition categories above and entered the **Warning Issuance Date/Time** data as desired, click **OK**. The name of your new **Time Window** will appear under the **Time Window** folder in the Study Pane, followed by an asterisk (*****) that indicates it is unsaved (Figure 10.7).

6. To save your study and your new **Time Window**, from the toolbar click 💾.

10.4   Save Time Window As

The **Save As** feature allows you to create a copy of a **Time Window** with a different name. This feature is useful if you want to create two similar time windows, but one with **Warning Issuance Time** calculated by impact area and the other by warning region, etc.
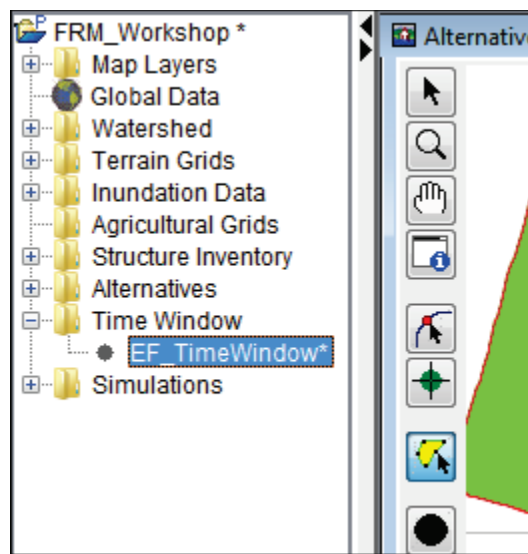
Figure 10.7 New time window under the Time Window folder of the Study Pane.

To create a copy of a time window using **Save As**:

1. Starting in the **Study Pane**, under the **Time Window** folder locate the name of the time window you wish to make a copy of.

2. From the shortcut menu, select **Save As** (Figure 10.8).
3. The **Save Time Window As** dialog box will appear. Enter a new name in the **Name** box and enter a description in the **Description** box.
4. Click **OK**. The **Save Time Window As** dialog box will close and the name of the new time window will appear under the **Time Window** folder of the **Study Pane**.
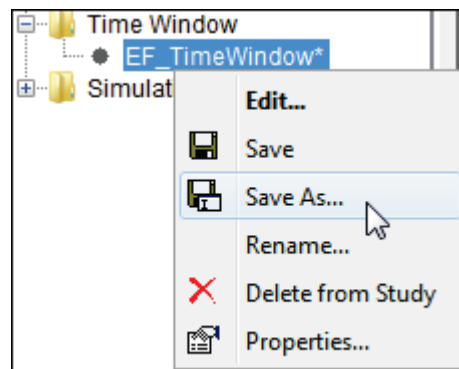
Figure 10.8 Time Window shortcut menu (Study Pane).

10.5   Delete a Time Window

Using the **Time Window** shortcut menu shown in Figure 10.8, users can also delete a **Time Window**.

To delete a time window from your study:
1. Starting in the **Study Pane**, under the **Time Window** folder right-click the name of the time window you wish to delete.
2. From the shortcut menu (Figure 10.8), select **Delete from Study**.
3. The **Confirm Manager Delete** dialog box will appear, notifying the user that the Time Window's file on the disk will also be deleted and prompting the user to confirm deletion. To delete, select **Yes**.
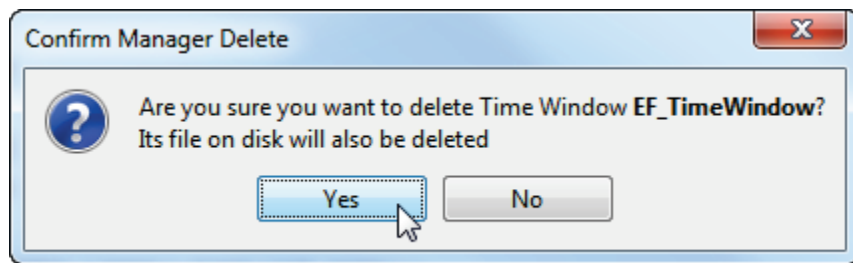
Figure 10.9 Confirm Manager Delete dialog box.

4. The time window will now be removed from your study and your hard drive.

10.6   Rename a Time Window

From the **Time Window shortcut menu** shown in Figure 10.8 (page 10-8), users can also rename a **Time Window**.

To rename a time window in HEC-FIA:

1. Starting in the **Study Pane**, under the **Time Window** folder locate the name of the time window you wish to rename.
2. From the shortcut menu, select **Rename** (see option in Figure 10.8).
3. The **Rename Time Window** dialog box will appear. Enter a new name for your time window and a short description.
4. Another dialog box will appear, prompting the user to confirm renaming of the file (Figure 10.10).
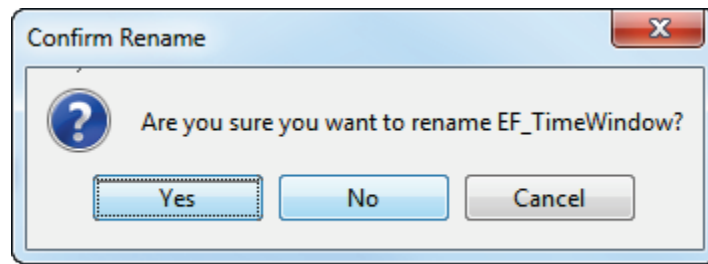
Figure 10.10 Confirm Rename dialog box.

5. Select **No** to cancel renaming and return to your study. Select **Cancel** to return to the **Rename Time Window** dialog and make a change in name or description. Select **Yes** to rename.
6. Your newly-renamed **Time Window** will appear under the **Time Window** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and your new **Time Window**, from the toolbar click .

Chapter 11

Simulations

Once you have created alternatives, you can begin to create **Simulations** that estimate consequences from flood events. Within the simulation setup the user can choose to do a life loss compute and/or compute agricultural damages. The user will also have the ability to define the population evacuation characteristics

11.4   Create Simulations

Two types of simulations can be performed in HEC-FIA: grid-based or time-series based. The following sections will cover the definition of each simulation type.

11.4.2   Create New Grid-Based Simulation

When defining a grid-based simulation, users will have the option to compute life-loss and agricultural damages based on the gridded data uploaded to their study *or* based on DSS hydrographs. In computing **Life Loss**, **Agriculture**, and **Evacuation Data**, users will have many options that will be explained in detail in their own sections. To that end, the next sections are broken down by option.

To get started creating a grid-based simulation:

1.   Starting in the **Study Pane**, right-click on the **Simulations** folder. From the shortcut menu, select **Create New Grid-Based Simulation** (Figure 11.1).
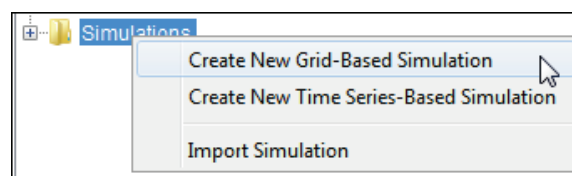
Figure 11.1 Simulation folder shortcut menu (Study Pane).

2.   The **Create New Simulations** dialog box will open. Enter a name (required) and a short description. Using the [...] browse button, select the directory in which your new simulation should be created. The study directory is the default directory (e.g. *C:\Users\ExampleUser\Documents\FIA\FRM_Workshop*).
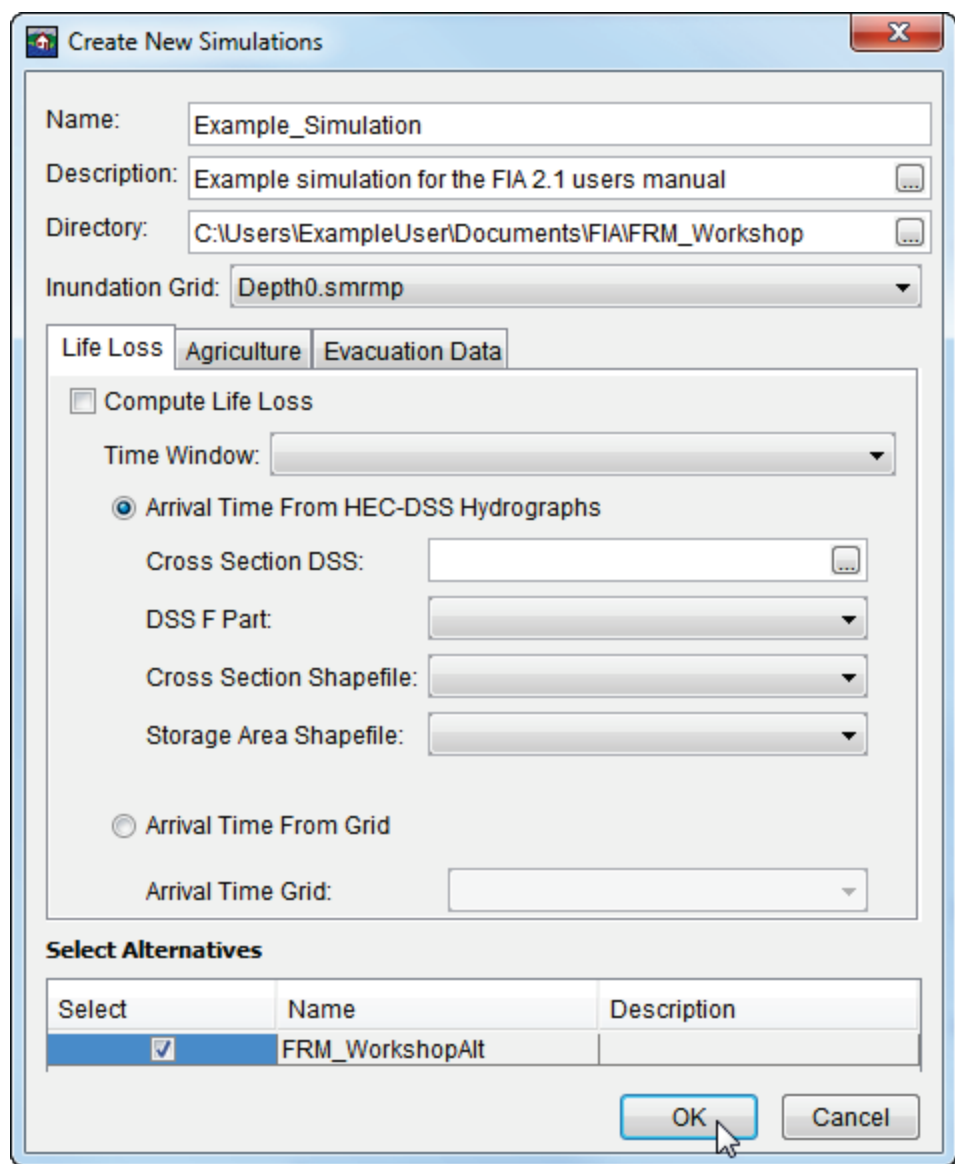
Figure 11.2 Create New Simulations dialog box (Grid-Based Simulation).

5. From the **Inundation Grid** dropdown list (Figure 11.2), select the inundation grid to use for this simulation. In the example, *Depth0.smrmp* is selected.

6. For now, skip over the three tabs that define the information necessary to compute **Life Loss**, **Agriculture**, and **Evacuation Data**. These tabs and their options will be covered in the following sections.

7. Under **Select Alternatives**, check the box next to the alternative(s) used in your simulation (Figure 11.2).

8. Click **OK**. The name of your new simulation will now
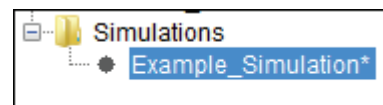
Figure 11.3 New simulation under Simulations folder (Study Pane).

appear in the **Study Pane** under the **Simulations** folder followed by an asterisk (*) that indicates it is unsaved (Figure 11.3).

9. To save your study and your new **Simulation**, from the toolbar click 🖫 .

10. If the user were to run the simulation at this point only urban damages would be computed. To compute **life loss** and **agricultural damages**, read the following sections and follow the instructions.

11.4.2.1 Compute Life Loss

To compute **life loss** in HEC-FIA, arrival time information is required to simulate the time that individuals will no longer be able to evacuate a structure (i.e. after the **non-evacuation depth** is achieved; see Define Safe Zone Fatality Rates, page 9-26). Users have the option of (1) computing **arrival time from HEC-DSS Hydrographs** (i.e. the interpolated time at which 2 ft depth or more has occurred between two hydrographs at cross sections) or (2) computing **arrival time from gridded data** (i.e. the time at which each grid reaches an inundation depth of 2 feet or more). Life loss computations can either be set up during the simulation creation process or entered into a previously created simulation. The example below will demonstrate entering life loss in the previously created simulation.

To enter the information for computing life loss damages:

1. In the **Study Pane**, under the **Simulations** folder right-click the name of the simulation you wish to edit. From the shortcut menu, select **Edit** (Figure 11.4).

2. The **Simulation Editor** will open. This is identical in functions to the **Create New Simulation** dialog (see Figure 11.2, page 11-2). The fields at the top (Name, Description, Inundation Grid) are already filled out and an alternative already selected. The Life Loss tab will be open by default.

Figure 11.4 Simulation shortcut menu (Study Pane).

3. Check the box next to **Compute Life Loss** (Figure 11.5).
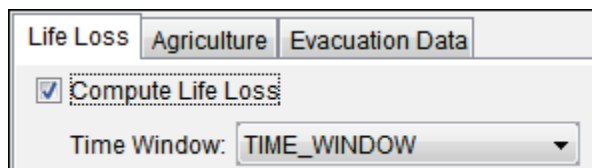
Figure 11.5 Simulation Editor—Life Loss tab.

4. From the **Time Window** dropdown list, select the appropriate **Time Window**. For creation of time windows, see Chapter 10, Section 10.2.

5. If you are computing arrival time information from **HEC-DSS Hydrographs**, follow the steps below. If you are using an **arrival time grid**, skip to step 6. See Appendix B, Section B.7.2 for preparation of arrival time information from HEC-DSS Hydrographs.

    a. Select **Arrival Time From HEC-DSS Hydrographs**
    b. For the field **Cross Section DSS**, use the ⊡ browse button to select the DSS file that contains the stage hydrograph for all cross sections in your study.
    c. From the **DSS F Part** dropdown list, select the flood event for which you want to calculate arrival time (e.g *Extreme_SDF*, for extreme standard design flood).
    d. From the **Cross Section Shapefile** dropdown list, select the shapefile that contains the cross sections for your study.
    e. From the **Storage Area Shapefile** list, select the shapefile with storage area information for your study.
    f. Skip to step 7.

6. To define **Arrival Time from Gridded** data:
    a. Select **Arrival Time From Grid**.
    b. From the **Arrival Time Grid** dropdown list, select the **arrival time grid** for life loss (e.g. *Life_Loss_Arrival.smrmp*) uploaded to your study (Figure 11.6). See Chapter 6, Section 6.5 for more on inundation data grids.
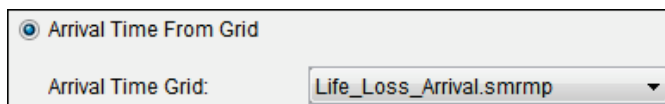
Figure 11.6 Life Loss Arrival Time from gridded data (Simulation Editor—Life Loss tab).

7. Verify that the correct alternatives are selected at the bottom of the **Simulation Editor**.
8. Click **OK**. The dialog will close and the name of your simulation will be followed by an asterisk (*) that indicates it is unsaved.
9. To save your study and your edited **Simulation**, from the toolbar click 🖫.

11.4.2.2 Compute Agriculture Damage

Arrival time information is also required to compute agricultural damages. This is either supplied in the form of an **arrival time grid**, or computed from a combination of **peak date and time** (i.e. the time at which the peak stage of the flood event has occurred) and the **crop loss function** you defined for your study (see Chapter 7, Section 7.5)

To enter the information for computing agriculture damages:

1. In the **Study Pane**, under the **Simulations** folder right-click the name of the simulation you wish to edit. From the shortcut menu, select **Edit** (Figure 11.4).
2. The **Simulation Editor** will be open. This is identical in functions to the **Create New Simulation** dialog (see Figure 11.11, page 11-10). The fields at the top (Name, Description, Inundation Grid) will be filled out and an alternative already selected.
3. Check the box next to **Compute Agriculture Damage** (see Figure 11.7).
4. If computing from an arrival time grid, skip to step 5. To compute arrival time from **peak stage** information:
    a. Select **Use Peak Date and Time** (see Figure 11.2, page 11-2).
    b. In the **Date** box, browse 🔲 to the calendar date on which the peak stage of your flood event will be reached or enter the

Figure 11.7 Compute agriculture damages from peak date and time (Simulation Editor—Agriculture tab).

   date in *ddmmm* format (Figure 11.7).
    c. In the **Time** box enter the 24 hour clock time of the peak stage (e.g. 0900).
    d. Skip to step 6.
5. To compute agricultural damages from an **arrival time grid**:
    a. Select **Use Arrival Time Grid**.
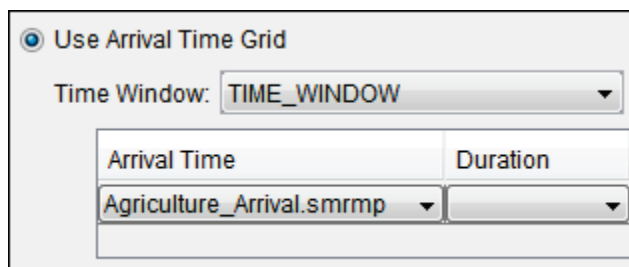
Figure 11.8 Compute agriculture damages from arrival time grid (Simulation Editor—Agriculture tab).

   b. From the **Time Window** dropdown list, select the appropriate **Time Window**.

   c. From the **Arrival Time Grid** dropdown list, select the **arrival time grid** for agricultural damages uploaded to your study (e.g. *Agriculture_Arrival.smrmp*) (Figure 11.8).

   d. If you have uploaded a **Duration Grid**, select it from the **Duration** dropdown list. For more on arrival time and duration grids, see Chapter 6, Section 6.3 and Appendix B.

7. Verify that the correct alternative(s) are selected at the bottom of the **Simulation Editor**.

8. Click **OK**. The dialog will close and the name of your simulation will be followed by an asterisk (*) that indicates it is unsaved.

9. To save your study and your edited **Simulation**, from the toolbar click ▣.

11.4.2.3 Enter Evacuation Data

Along with the arrival time information you entered in the Compute Life Loss section (page 11-3), the data entered under the **Evacuation Data** tab of the **Simulation Editor** is also required to compute life loss.

Within this tab, users must either define manually the total **Evacuation Time** (i.e. the time in minutes an individual would need to get from the structure to safety) for each **Warning Region, Individual Structure,** or for your entire inventory or **Let HEC-FIA Compute** evacuation time by entering an estimated **Evacuation Velocity** (i.e., the estimated velocity of evacuees during the evacuation process). Because of the potential arduousness and/or inaccuracy of the former, the option **Let HEC-FIA Compute** is recommended. If **Let HEC-FIA Compute** is chosen as the evacuation option evacuees will travel at a user defined velocity in a straight line to the closest edge of the hazard area shapefile. Thus, entering evacuation data by this option will be covered first, with the other time definition methods following. Though a user can calculate independent

times for each structure for evacuation as a preprocess using an evacuation model, it is unlikely users will do this. Given that, and the fact that FIA is making general assumptions about the evacuation using a shortest distance and a nominal evacuation speed in the **Let HEC-FIA Compute** option, it is important to note that if evacuation is a primary risk driver in the area being studied, the user should consider evaluating the life loss with **LifeSim**.

To let HEC-FIA compute evacuation time:

1. In the **Study Pane**, under the **Simulations** folder right-click the name of the simulation for which you wish to enter evacuation data. From the shortcut menu, select **Edit**.
2. The **Simulation** editor will be evoked. This is identical in functions to the **Create New Simulation** dialog (see Figure 11.2, page 11-2). The fields at the top (Name, Description, Inundation Grid) will be filled out and an alternative already selected.
3. Select the **Evacuation Data** tab of the **Simulation Editor** (Figure 11.9).

Figure 11.9 Let HEC-FIA compute evacuation time (Simulation Editor—Evacuation Data tab).

4. From the **Define Time By** dropdown list, select **Let HEC-FIA Compute**.
5. Select the Hazard Area Shapefile uploaded to your study from the **Select Hazard Area Shapefile** dropdown list. See Appendix B, Section B.6 for preparation of hazard area shapefiles; see Chapter 3, Section 3.2 for uploading shapefiles to a study.
6. In the **Nominal Evacuation Velocity** field, enter an estimate of the evacuation velocity in mph. In Figure 11.9, 3 mph has been entered to reflect a moderate walking speed.
7. At the bottom of the **Simulation Editor** (see Figure 11.2, page 11-2), click **OK**. The dialog will close and the name of your simulation will be followed by an asterisk (*) that indicates it is unsaved.
8. To save your study and your edited **Simulation**, from the toolbar click ⊞.

To manually define evacuation time:

1. In the **Study Pane**, under the **Simulations** folder right-click the name of the simulation for which you wish to enter evacuation data. From the shortcut menu, select **Edit**.
2. The **Simulation** editor will be evoked. The functions of the editor are identical to that of the **Create New Simulation** dialog (see Figure 11.2, page 11-2). The fields at the top (Name, Description, Inundation Grid) will be filled out and an alternative already selected.
3. Select the **Evacuation Data** tab of the **Simulation Editor** (Figure 11.10).

Figure 11.10 Define evacuation time by individual structure (Simulation Editor—Evacuation Data tab).

4. From the **Define Time By** dropdown list, select the unit by which you would like to define time. In the example in Figure 11.10, **Individual Structure** has been selected so that a different **Evacuation Time** (i.e. the time it takes for an occupant of the structure to get to safety) can be entered for *each structure in the structure inventory*. Users can also choose to define evacuation time by **Warning Region**, **Impact Area**, or define one evacuation time for their entire study (**All Structures**).
5. Each option will evoke a table with the name of the unit you selected in step 4 and a field to define the evacuation time in minutes. In the example in Figure 11.10, the name of each individual structure appears to the left, and different evacuation times have been entered in the table.
6. Since Evacuation Time can only be defined by one unit in each simulation, users may create multiple simulations to define evacuation time by different units, if desired (e.g. one simulation has evacuation time defined by **Warning Region**, and another by **Impact Area**).
7. At the bottom of the **Simulation Editor** (see Figure 11.2, page 11-2), click **OK**.
8. The dialog will close and the name of your simulation will be followed by an asterisk (*) that indicates it is unsaved.

9. To save your study and your edited **Simulation**, from the toolbar click ![save icon].

Now that you've created a simulation and entered the necessary data for life loss and agricultural damage compute, you are ready to begin running simulations. To run a simulation, see Section 11.7, page 11-12.

11.4.3   Create New Time-Series-Based Simulation

Before you can create a simulation from time-series HEC-DSS hydrographs you must create a **Time-Series-Based Alternative** (see Chapter 9, Section 9.3) and a **Time Window** (see Chapter 10).

To define a new time-series-based simulation:

1. Starting in the **Study Pane**, right-click on the **Simulations** folder. From the shortcut menu (Figure 11.1), select **Create New Time Series-Based Simulation**.
2. The **Create New Time Series Simulation** dialog box will open (Figure 11.11). Enter a name (required) and a short description. Using the ⬚ browse button, select the directory in which your new simulation should be created.
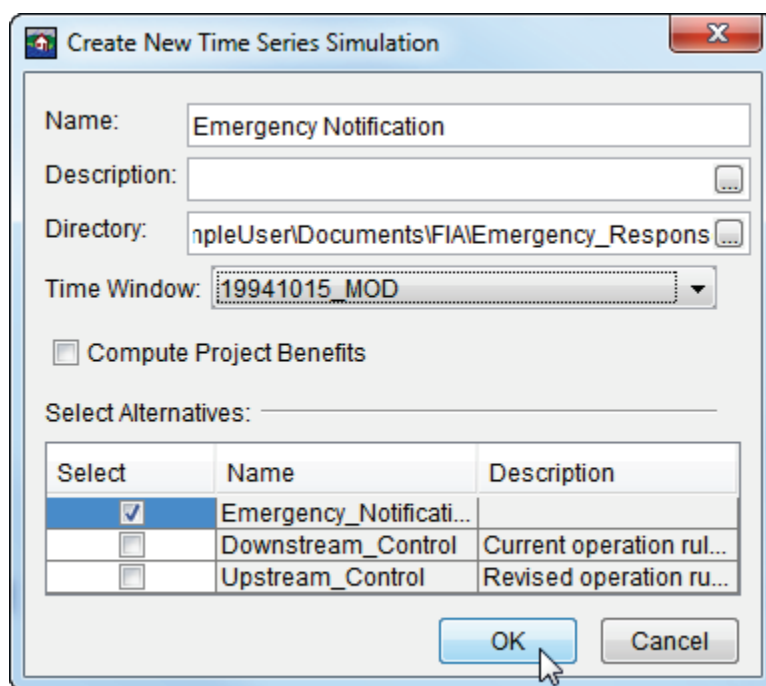
Figure 11.11 Create New Time Series Simulation dialog box.

3. From the **Time Window** dropdown list, select the appropriate **Time Window** for your simulation.
4. Check the **Compute Project Benefits** box if you want FIA to compute project benefits (see Chapter 9, Section 9.9.3 for an explanation of project benefits).
5. The **Alternatives** box displays the alternatives that have been created for the watershed. To associate an alternative with a simulation, in the **Select** column of the table, check the box next to the alternative you wish to select.
6. In Figure 11.11, one alternative, *Emergency_Notification* has been chosen for the *Emergency Notification* simulation.

7. Click **OK**. The name of your new **Simulation** will appear under the **Simulations** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and your new **Simulation**, from the toolbar click ⊞ .
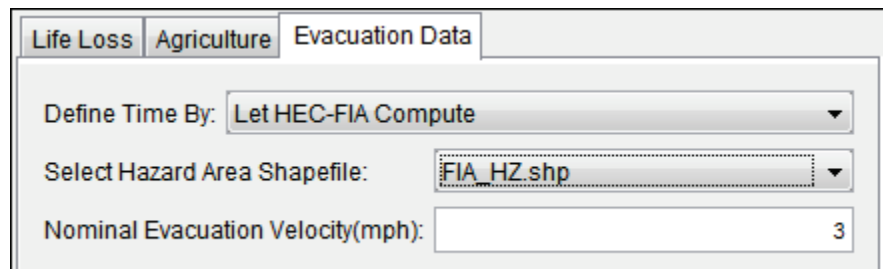
11.5 Import Simulation

Users can import simulations from another FIA study using the shortcut menu for simulations (Figure 11.1):

1. Starting in the **Study Pane**, right click on the **Simulation** folder and from the shortcut menu select **Import Simulation**.
2. An **Open** browser will appear. Browse to the folder that contains the project (*.prj) file with the simulation you want to import. Select the project file and click **Open**.
3. The **Select** dialog box (Figure 11.12) will appear.
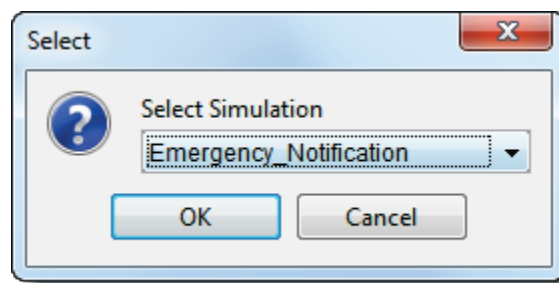
Figure 11.12 Select dialog box.

4. The **Select Simulation** dropdown list shows the simulations within the project file you have opened. Select the simulation you would like to import and click **OK**.
5. A progress dialog box will appear and close automatically when the import is complete. The newly imported simulation will appear under the **Simulation** folder in the **Study Pane**.

11.6 Edit Simulation

Users may edit simulations they have created using the **Simulations Editor**. As stated in Section 11.4.2, the **Simulations Editor** is identical in functions to the **Create New Simulations** dialog boxes.

To edit a simulation:
1. Starting in the **Study Pane**, locate the name of the simulation you wish to edit and right-click. From the shortcut menu, select **Edit** (see Figure 11.4, page 11-3).
2. The simulation editor will be evoked. As they are identical in function to the dialog boxes used to create simulations, the editor is different for **Time-Series-Based** and **Grid-Based** simulations.

3. Make the desired changes according to the instructions in Section 11.4.2 (page 11-1).
4. Once all changes have been made, click **OK**.
6. Your simulation will appear under the **Simulations** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and the new edits to your simulation, from the toolbar click 💾.

11.7   Run Simulations

Now that you have created and defined a simulation, you are ready to run simulations.

11.7.2   Run Single Simulation

To compute a single simulation in FIA:

1. Starting in the **Study Pane**, locate the name of the simulation you wish to compute and right-click. From the shortcut menu, select **Compute**.
2. A dialog box will appear displaying the progress of your compute (Figure 11.13). For a single simulation, this may take a few minutes. Once it is finished, select **Close**.
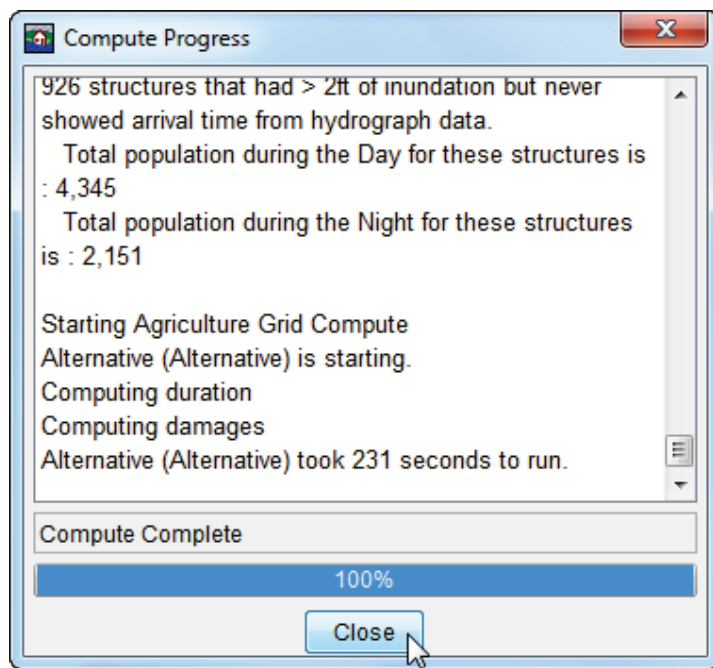
Figure 11.13 Compute Progress dialog box

3. The **Content Pane** will display the last run date of your simulation as well as the alternatives within it. For viewing simulation results, see Chapter 12.

11.7.3   Run Multiple Simulations

To run multiple simulations in FIA:

1. From the **Menu Bar**, select **Simulation**➜**Compute Manager** (Figure 11.14).
2. The **Compute manager** will open. This contains a list of the simulations created in your study.
3. Select the simulations you wish to run, and press **Compute** (Figure 11.15).
4. A dialog box will appear, showing compute progress for each simulation. This may take some time depending on the number of simulations you are running. Once the compute process is finished, click **Close** (Figure 11.13).
5. To view simulation results, see Chapter 12.
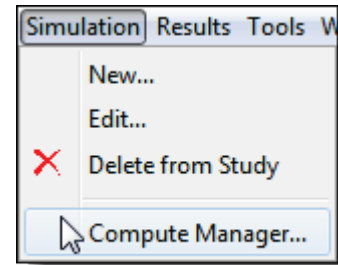
Figure 11.14 Simulation Menu (Menu Bar).
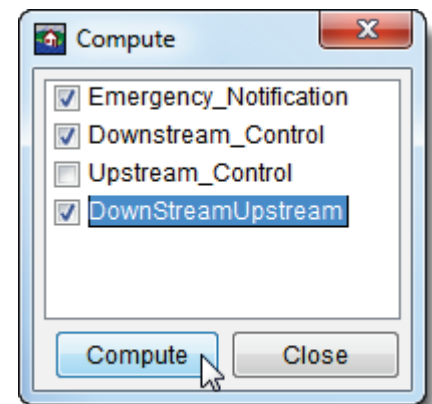
Figure 11.15 Compute manager.

11.8   Rename Simulation

To rename a simulation:

1. Starting in the **Study Pane**, under the **Simulations** folder right-click the name of the simulation you wish to rename. From the shortcut menu, select **Rename**.
2. The **Rename Simulation** dialog box will appear. Enter a new name for your simulation and a short description.
3. Another dialog box will appear, prompting the user to confirm renaming of the file (Figure 11.16).
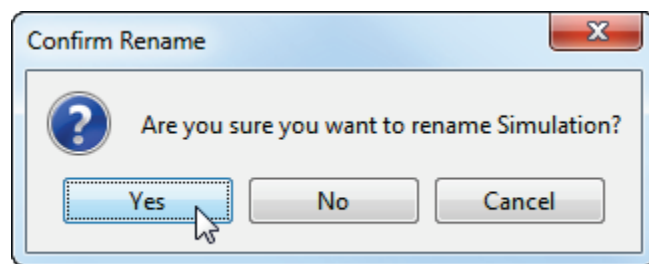
Figure 11.16 Confirm Rename dialog box.

4. Select **Yes** to rename. Select **Cancel** to return to the **Rename Time Window** dialog and make a change in name or description.

5. Your newly-renamed simulation will appear under the **Simulations** folder in the **Study Pane**, followed by an asterisk (*) that indicates it is unsaved.
6. To save your study and your newly-renamed simulation, from the toolbar click 🖫.

11.9 Delete Simulation

To delete a time window from your study:

1. Starting in the **Study Pane**, under the **Simulations** folder right-click the name of the simulation you wish to delete.
2. From the shortcut menu, select **Delete from Study**.
3. The **Confirm Manager Delete** dialog box will appear (Figure 11.17), notifying the user that the simulation will be deleted from the hard drive and prompting the user to confirm deletion. To delete, select **Yes**.
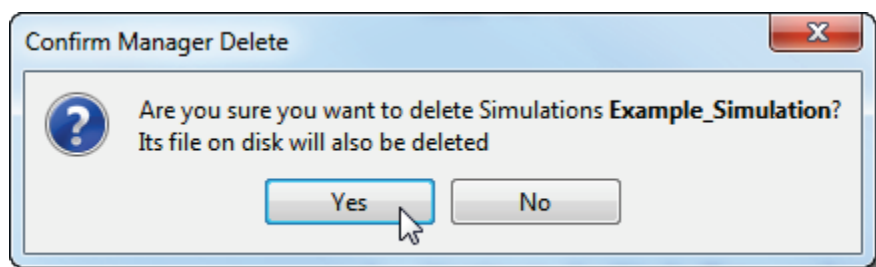
Figure 11.17 Confirm Manager Delete dialog box.

4. The simulation will now be removed from your study and your hard drive.

Chapter 12

Results

After a simulation has been executed, results are available in several different reports: Individual Structure Damage, Aggregated Structure Damage, Agriculture Damage, Detailed Life Loss, and Consequence Summary.

12.1   Individual Structure Damage Report

The **Individual Structure Damage Report** shows detailed damage results for each structure in your **Structure Inventory**.

12.1.1   View Individual Structure Damage Report

To view individual structure damage results:

1. Select a computed simulation under the **Simulation** folder of the **Study Pane**. The **Content Pane** will display information from the simulation run. From the **Content Pane**, under the **Alternatives** folder right-click on the name of the alternative for which you want to view results. From the shortcut menu, select **Individual Structure Damage Report** (**Figure 12.1**).
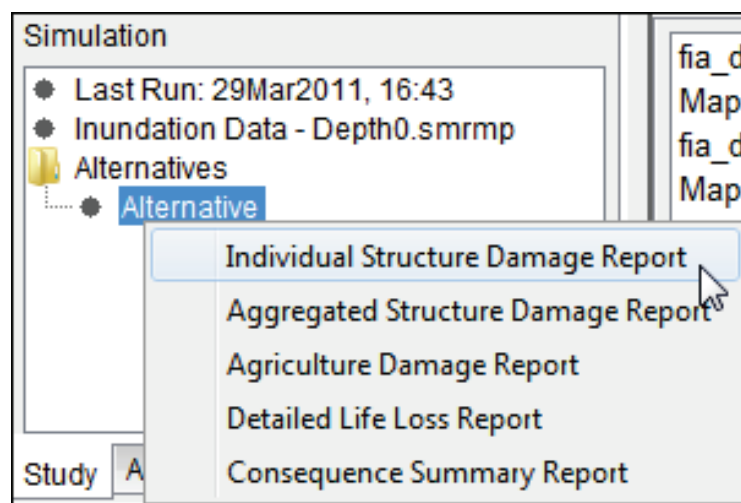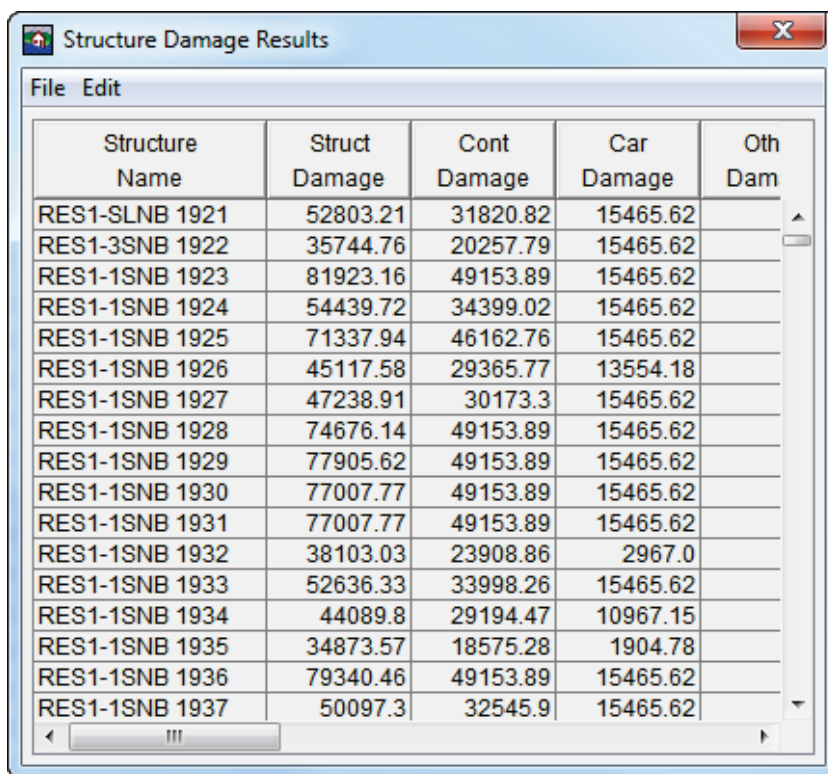
Figure 12.1 Alternative shortcut menu (Content Pane).

2. The **Structure Damage Results** dialog box will open (**Figure 12.2**).

Figure 12.2 Individual Structure Damage Report.

3. In the column **Structure Name** is a list of every structure in your structure inventory. To the right, results in 14 categories are listed in the columns. Damage results include structure, content, car, and other (user-specified) damage as well as total damages for each structure. Inundation results include depth of flooding, population at risk (day and night), warning time, percent not mobilized, percent caught evacuating, percent clear, and loss of life (day and night).

4. When done reviewing click **Close**. The **Individual Structure Damage Report** dialog box will close.

12.1.2   Print Individual Structure Damage Report

To print the **Individual Structure Damage Report**:

1. Open the **Individual Structure Damage Report**.
2. Right-click on any cell. From the shortcut menu, select **Print** (**Figure 12.3**).
3. The **Print Properties** dialog box will open, showing the print options for the table (**Figure 12.4**).

Figure 12.3 Individual Structure Damage Report shortcut menu.
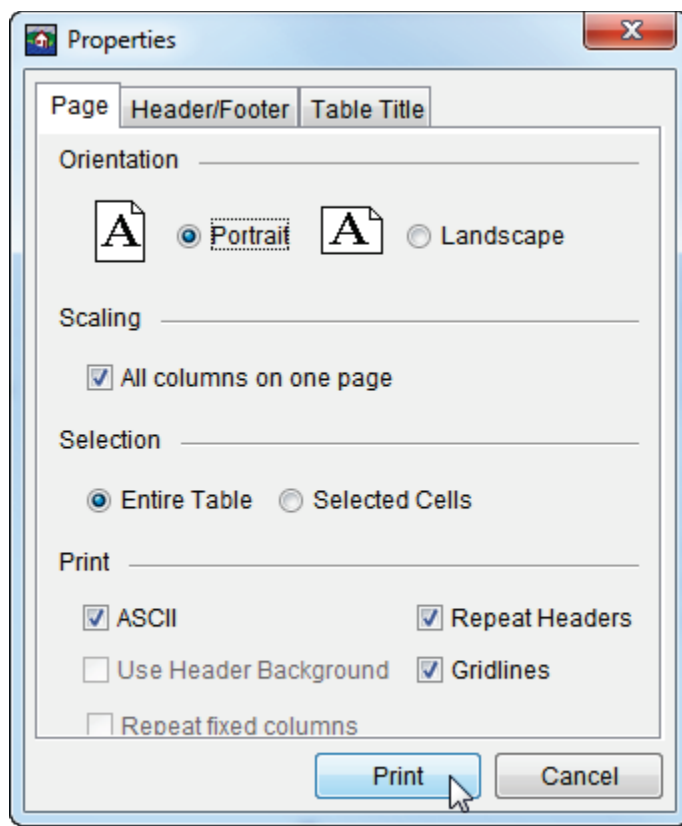
Figure 12.4 Print Properties (Individual Structure Damage Report).

4.   From the **Page** tab (default worksheet), you can set the orientation (default is Portrait), make the report columns fit on one page, print the entire table (default) or print selected cells, repeat headers on subsequent pages, and display gridlines. By default the report print in an ASCII format, if you want a screen image of the report, unclick ASCII and the report will print with any shading/color formatting that you see on the report.

5.   The **Header/Footer** tab allows you enter a header or footer for the report.

6.   The **Table Title** tab allows you to change the default title of the report, and to determine whether you want the title repeated on subsequent pages.

7.   After you have everything set, click **Print**. Your default system print settings dialog box will appear. Select the printer, and any other options you wish to set, click **OK**. The report will then be printed on the selected printer.

12.1.3   Export Individual Structure Damage Report to File

Users may export most reports as text (.txt) files.

To export an Individual Structure Damage report to file:

1.   Open the Individual Structure Damage Report.

2. Right-click on any cell. From the shortcut menu, select **Export** (see **Figure 12.3**).

3. The **Table Export Options** dialog box will open (**Figure 12.5**).
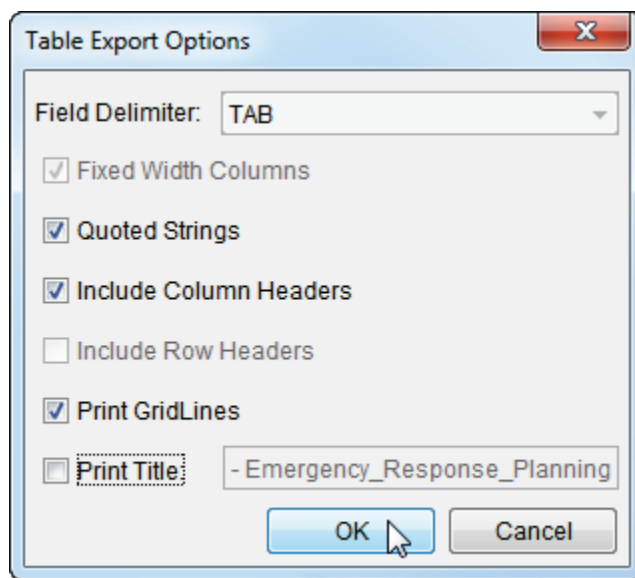
Figure 12.5 Table Export Options dialog box.

4. Users can specify the delimiter to be used. The default is **TAB**. Your choices are **TAB, SPACE, COMMA**, or **COLON**.

5. You can also include the column headers of the report (default), include gridlines, and include the title of the report.

6. Once the options are set, click **OK**, the system **Save Browser** will open.

7. A file browser will open. Browse the location you would like to save your report.

8. Enter a name for your report and the .txt file extension (e.g. *FRM_Workshop_ISD_Report.txt*) and select **Save**.

10. The dialog will close and the report will be saved to your hard drive.

12.2   Aggregated Structure Damage Report

The **Aggregated Structure Damage Report** shows a summary of urban damage and structures inundated. Users can view, print, or export aggregated structure damage results.

12.2.1   View Aggregated Structure Damage Report

To view aggregated structure damage results:

1. Select a computed simulation under the **Simulation** folder of the **Study Pane**. The **Content Pane** will display information from the simulation run. From the **Content Pane**, under the **Alternatives**

folder right-click on the name of the alternative for which you want to view results. From the shortcut menu (**Figure 12.1**, page 12-1), select **Aggregated Structure Damage Report**.

2. The **Aggregated Structure Damage Report** dialog box will open (**Figure 12.6**).
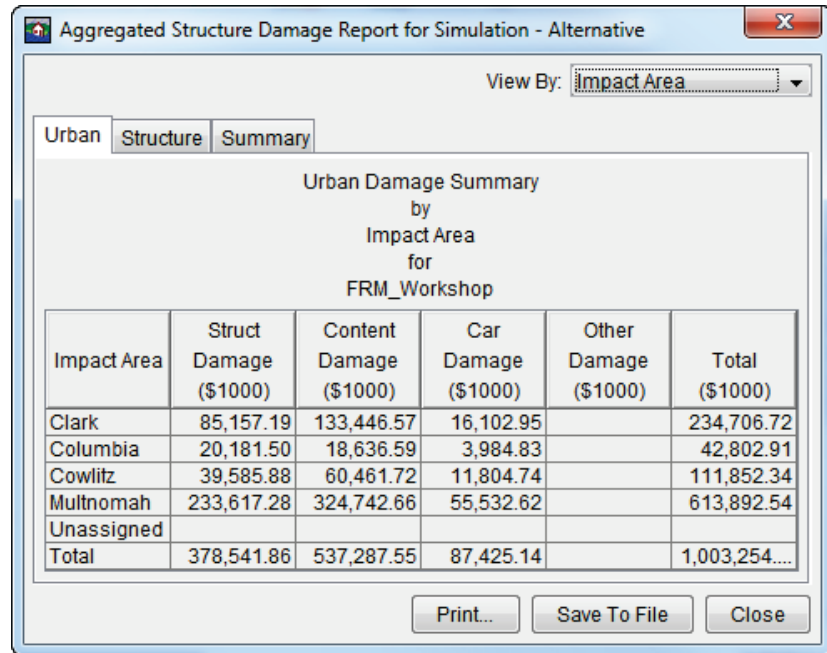
Figure 12.6 Aggregated Structure Damage Report.

3. The Aggregated Structure Damage Report will display urban damage results by Impact Area by default. To toggle the **View By** boundary, from the **View By** dropdown list select the boundary for which you wish to see results (**Figure 12.7**). You will only be able to view reports by the boundaries you have defined in the **Global Data Editor** (i.e. if you have not defined

Figure 12.7 Toggling view by boundaries (Aggregated Structure Damage Report).

Congressional District boundaries, you will not be able to view damages by that boundary). See Chapter 4, Section 4.3 for setting up boundaries in the **Global Data Editor**.

4. There are three different tabs in the Aggregated Structure Damage Report:

    a. In the **Urban Damage Summary** (**Figure 12.6**), users can view Structure Damage, Content Damage, Car Damage, and Total Damages by the boundaries in step 3.

b. In the **Structures Flooded Summary**, users can view the number of Commercial, Industrial, Public, and Residential Structures flooded, as well as total number of structures flooded in each boundary area. In the example in **Figure 12.8**, the Multnomah Impact Area experienced the most structure flooded with 2,691 structures flooded.

Figure 12.8 Structures Flooded Summary (Aggregated Structure Damage Report).

c. Under the **Summary** tab, users can view consequence summaries. Summary categories include **Total Urban Damages**, **Total Crop Damages**, **Total Damages**, **Total Acreage Flooded**, **Total Number of Structures Flooded**, **Total Population Impacted**, and **Total Population over 65 Impacted**.

4. When done viewing results, click **Close**. The **Aggregated Structure Damage Report** dialog box will close.

12.2.2   Print Aggregated Structure Damage Report

To print the **Aggregated Structure Damage Report**:

1. First, open the **Aggregated Structure Damage Report** (Section 12.2.1).
2. At the bottom of the **Aggregated Structure Damage** dialog (see **Figure 12.6**, page 12-5), select **Print**.
3. The **Print Properties** dialog box (see **Figure 12.4**, page 12-3) will open.
4. Make the same adjustments for printing as described in Section 12.1.2 (page 12-2).
5. After all options are set, click **Print**. Your default system print settings dialog box will appear. Select the printer, and any other options you wish to set, click **OK**. The report will then be printed on the selected printer.

12.2.3   Export Aggregated Structure Damage Report to File

Users may export most reports as binary (ASCII) or text (.txt) files.

To save the Aggregated Structure Damage report:
1. First, open the Aggregated Structure Damage Report (step 1 in the previous section 12.2.1, page 12-4).
2. At the bottom of the Aggregated Structure Damage dialog (see **Figure 12.6**, page 12-5), select **Save To File**.
3. The **Table Export Options** dialog box will open (see **Figure 12.5**, page 12-4).
4. You can specify the delimiter to be used. The default is **TAB**. Your choices are **TAB, SPACE, COMMA**, or **COLON**.
5. You can also include the column headers of the report (default), include gridlines, and include the title of the report.
6. Once you are set, click **OK**, the system **Save Browser** will open.
7. A file browser will open. Browse the location you would like to save your report.
8. Enter a name for your report and the .txt file extension (e.g. *FRM_Workshop_ASD_Report.txt*) and select **Save**.
9. The dialog will close and the report will be saved to your hard drive.

12.3   Agriculture Damage Report

In the Agriculture Damage Report, users can view agriculture in thousands of dollars as well as agricultural area flooded. Users can view, print, or export the report to file.

12.3.1   View Agriculture Damage Report

To view the agriculture damage results:

1. Information from computed simulations is contained in the **Content Pane**. From the **Content Pane**, under the **Alternatives** folder right-click on the name of the alternative for which you want to view results. From the shortcut menu (**Figure 12.1**, page 12-1), select **Agriculture Damage Report**.
2. The **Agriculture Damage Report** dialog box will open (**Figure 12.9**).
3. From the dropdown list in the upper right-hand corner of the dialog box, select the boundary for which you wish to see results. You will only be able to view reports by the boundaries you have defined in the **Global Data Editor** (see Chapter 4, Section 4.3).
4. The report will display damage in thousands of dollars and total area flooded for each boundary selected in step 3. The example shows the boundary **Impact Area** selected, and damage and area flooded for each impact area in the study.
5. When finished viewing the report, click **Close** to exit.

Figure 12.9 Agriculture Damage Report.

12.3.2   Print Agriculture Damage Report

To print the **Agriculture Damage Report**:

1. Open the **Agriculture Damage Report** (see Section 12.3.1).
2. From the **File** menu, select **Print**.
3. The **Print Properties** dialog box will open, showing the print options for the table (see **Figure 12.4**, page 12-3).
4. Add a header and footer for your report if desired under the **Header/Footer** tab. Add a title for your table if desired under the **Table Title** tab. Toggle other options if desired.
5. At the bottom of the dialog, click **Print**.
6. Your default system print settings dialog box will appear. Select the printer, and any other options you wish to set, click **OK**. The report will then be printed on the selected printer.

12.3.3   Export Agriculture Damage Report to File

Users may export most reports as text (.txt) files.

To save the Agriculture Damage report:

1. Open the Agriculture Damage Report (see Section 12.3.1).
2. From the menu bar of the Agriculture Damage dialog, select **File→ Export**.
3. The **Table Export Options** dialog box will open (see **Figure 12.5**, page 12-4).
4. You can specify the delimiter to be used. The default is **TAB**. Your choices are **TAB, SPACE, COMMA**, or **COLON**.

5. You can also include the column headers of the report (default), include gridlines, and include the title of the report.
6. Once you are set, click **OK**, the system **Save Browser** will open.
7. A file browser will open. Browse the location you would like to save your report.
8. Enter a name for your report and the .txt file extension (e.g. *FRM_Workshop_ASD_Report.txt*) and select **Save**.
9. The dialog will close and the report will be saved to your hard drive.

12.4   Detailed Life Loss Report

In the Detailed Life Loss Report (**Figure 12.10**), users view the results of life loss simulation in their study, which includes results on life loss, warning diffusion, mobilization, and evacuation. The report can be printed, but cannot be exported to file.

Figure 12.10 Detailed Life Loss (Life Sim) Report.

The report shows results from the **Simplified Life Sim** engine that calculates life loss as number of people (**Num. People**) and **Percentage** of population for those under or over sixty-five (65) years of age at day and night. The report is provided for users to pinpoint how and where life loss happened during their simulation and evaluate the driving factors of life loss. The report can also be useful for troubleshooting the simulation results (e.g. if none of a population received a warning, parameters for warning diffusion may be incorrectly set; see Chapter 9, Section 9.10.6 for more on life loss parameters).

The detailed results are contained in a file tree structure. **PAR** stands for **Population at Risk**, i.e. the number of people put in danger by the modeled flood event. The results contained in the following folders represent portions of the population at risk.

The **Received Warning** folder gives life loss results for those of the PAR who received and understood a warning message (e.g. an evacuation order after a dam failure). Of those who received a warning, results for those who were **Mobilized** show the number/percentage of population that were **Cleared** (i.e. those who were able to evacuate outside the hazard area boundary) and those who were **Caught** (those who left their structures but were unable to reach the hazard area before inundation). Of those who were Cleared, the report shows how many **Started** Inundated, that is, how many individuals were able to clear an area where flood waters had already arrived. Of those who were **Caught**, the report shows how many **Survived** and how many lost their lives (**Lost Life**). The **Not Mobilized** category shows how many of those who received a warning message did not evacuate.

The **Not Mobilized** folder shows the total number of the population at risk who did not evacuate the area – some of whom were **Warned** and some of whom were **Not Warned**. The **In Buildings** folder contains subfolders for one, two, and three story structures and how many **Survived** and **Lost Life** in each.

Other folders show how many individuals **Started Inundated** in the initial condition of the simulation and summaries of those who were cleared and those who lost their lives.

12.4.1   View Detailed Life Loss Report

To view life loss results:

1. Select a computed simulation under the **Simulation** folder of the **Study Pane**. The **Content Pane** will display information from the simulation run. From the **Content Pane**, under the **Alternatives** folder right-click on the name of the alternative for which you want to view results. From the shortcut menu (**Figure 12.1**, page 12-1), select **Detailed Life Loss Report**.
2. The **Life Sim Report** dialog box will open (**Figure 12.10**).
3. This report provides details for life loss parameters. From the **View By** list, details can be viewed by Total or by Impact Area.
4. Click ![icon] to add decimal places. Click ![icon] to remove decimal places.
5. When done reviewing click **Close**. The **Life Sim Report** dialog box will close.

12.4.2   Print Detailed Life Loss Report

To print the **Detailed Life Loss Report**:

1. Open the **Detailed Life Loss Report** (see Section 12.4.1).
2. To print, at the bottom of the dialog select **Print** (**Figure 12.10**).
3. The **Print Properties** dialog box will open, which is similar to that shown in **Figure 12.4** (page 12-3), only the **Print** button has been replaced by a **Preview** button.
4. Add a header and footer for your report if desired under the **Header/Footer** tab. Add a title for your table if desired under the **Table Title** tab. Toggle other options if desired.
5. At the bottom of the dialog, click **Preview**.
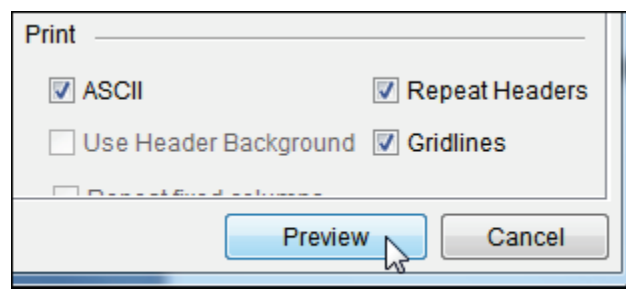
Figure 12.11 Print Properties (Detailed Life Loss Report).

6. The **Print Preview** dialog box will open, showing a preview of your printed report (**Figure 12.12**).
7. At the top of the dialog, select **Print**.
8. Your default system print settings dialog box will appear. Select the printer and any other options you wish to set, and then click **OK**. The report will then be printed on the selected printer.

12.5   Consequence Summary Report

In the **Consequence Summary Report**, users view results for Total Damages (including urban and crop damages), Total Acreage Flooded, Total Number of Structures Flooded, the population at risk (PAR) at day and night, and life loss (day and night). The report can be printed and exported to file.

12.5.1   View Consequence Summary Report

To view consequence summary results:

1. Select a computed simulation under the **Simulation** folder of the **Study Pane**. The **Content Pane** will display information from the simulation run. From the **Content Pane**, under the **Alternatives** folder right-click on the name of the alternative for which you want
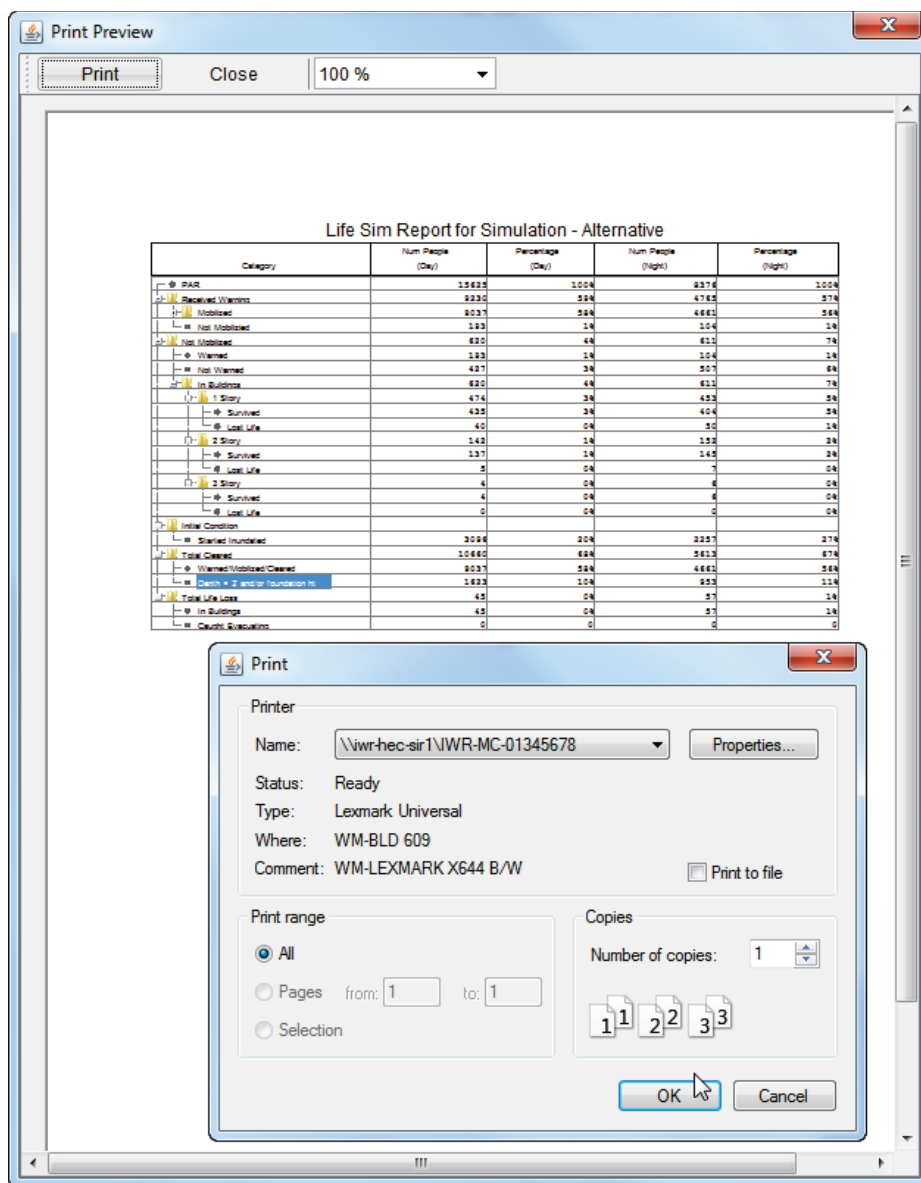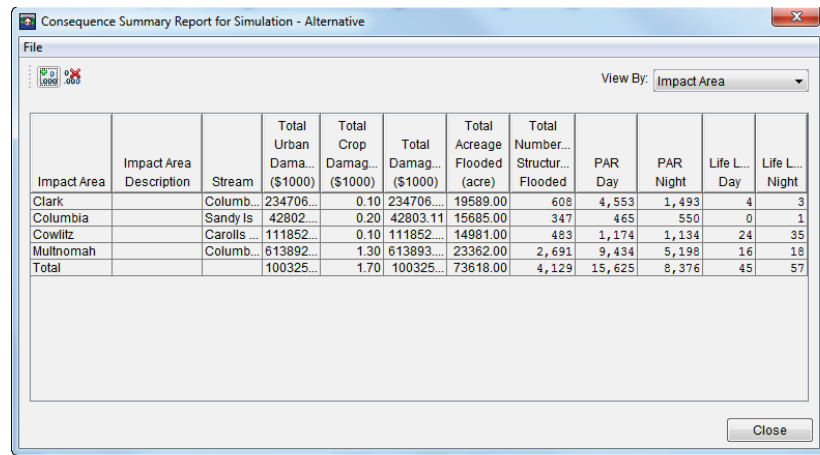
Figure 12.12 Background: Print Preview; foreground: system print settings dialog box (Detailed Life Loss Report).

> to view results. From the shortcut menu (**Figure 12.1**, page 12-1), select **Consequence Summary Report**.

2. The **Consequence Summary Report** dialog box will open (**Figure 12.13**).

3. This report shows **Total Damages** (including urban and crop damages), **Total Acreage Flooded**, **Total Number of Structures Flooded**, the population at risk (**PAR**) at day and night, and **life loss** (day and night). From the **View By** dropdown list, details can be viewed by **Total** or by **Impact Area**.

4. Click [icon] to add decimal places. Click [icon] to remove decimal places.

5. When done reviewing click **Close**.

Impact Area	Impact Area Description	Stream	Total Urban Dama... ($1000)	Total Crop Damag... ($1000)	Total Damag... ($1000)	Total Acreage Flooded (acre)	Total Number... Structur... Flooded	PAR Day	PAR Night	Life L... Day	Life L... Night
Clark		Columb...	234706...	0.10	234706...	19589.00	608	4,553	1,493	4	3
Columbia		Sandy Is	42802...	0.20	42803.11	15685.00	347	465	550	0	1
Cowlitz		Carolls ...	111852...	0.10	111852...	14981.00	483	1,174	1,134	24	35
Multnomah		Columb...	613892...	1.30	613893...	23362.00	2,691	9,434	5,198	16	18
Total			100325...	1.70	100325...	73618.00	4,129	15,625	8,376	45	57

Figure 12.13 The Consequence Summary Report

12.5.2 Print Consequence Summary Report

To print the **Consequence Summary Report**:
1. Open the Consequence Summary Report (see Section 12.5.1,).
2. From the **File** menu, select **Print**.
3. The **Print Properties** dialog box will open, showing the print options for the table (see **Figure 12.4**, page 12-3).
4. Add a header and footer for your report under the **Header/Footer** tab. Add a title for your table if desired under the **Table Title** tab. Toggle other options if desired.
5. At the bottom of the dialog, click **Print**.
6. A typical Windows printer settings dialog will appear. Click **OK** to print the Consequence Summary Report.

12.5.3 Export Consequence Summary Report to File

To export the **Consequence Summary Report** to file:
1. Open the **Consequence Summary Report** (see Section 12.3.1).
2. From the menu bar of the Consequence Summary dialog, select **File→Export**.
3. The **Table Export Options** dialog box will open (see **Figure 12.5**, page 12-4).
4. You can specify the delimiter to be used, the default is **TAB**. Your choices are **TAB, SPACE, COMMA**, or **COLON**.
5. You can also include the column headers of the report (default), include gridlines, and include the title of the report.
6. Once you are set, click **OK**, the system **Save Browser** will open.
7. A file browser will open. Browse the location you would like to save your report.
8. Enter a name for your report and the .txt file extension (e.g. *FRM_Workshop_ASD_Report.txt*) and select **Save**.

9. The dialog will close and the report will be saved to your hard drive.

12.6   Project Benefits Results

FIA computes individual project benefits for reservoirs and levees in the watershed as outlined in Appendix C. These individual project benefits can then be reported for any impact area or any other geographic or political boundary specified for an impact area. If the individual project benefits are to be reported for an impact area, you must first select the compute or assign project benefits button when defining alternatives in the **Impact Area Editor** (see Chapter 9, Section 9.7.3).

From the **Project Benefits Report** dialog box (**Figure 12.15**) you can review the results for the selected alternative. There are four reports available that record urban damages, structures impacted, population impact, and agricultural damages.

12.6.1   Urban Damage Project Benefits Report

To view **Urban Damage Project Benefits Report**:

1. From the **Menu Bar**, select **Results➔ Project Benefits Report** (**Figure 12.14**).
2. A dialog box will appear. From the **Simulation** dropdown menu, select the simulation for which you would like to view project benefits results.

Figure 12.14 Results menu (Menu Bar).

3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.
4. Select the alternative(s) for which you would like to view impact response results and press **OK**.
5. The **Project Benefits Report** dialog box will appear (**Figure 12.15**), by default on the **Urban Damages** report. This shows the total damage incurred in your simulation with and without the projects (reservoirs and levees) you have created. Under the **Damage Categories** column, the damage categories you created in Chapter 4, Section 4.4 will appear.

Figure 12.15 Project Benefits Report

6. From the **View By** list, results can be viewed by boundary.
7. The report contains results for the selected alternative. Results for the with- and without-project damages are reported by impact area and damage categories. At the bottom, users can view the total damages with and without projects.
8. If the selected alternative included projects in its definition, then results will be reported in the next two areas of the report.
9. When finished viewing the report, click **Close**. The **Project Benefits Report** dialog box will close.

12.6.2   Structures Impacted Project Benefits Report

To view **Structures Impacted Project Benefits Report**:

1. From the **Menu Bar**, select **Results➤Project Benefits Report** (**Figure 12.14**).
2. A dialog box will appear. From the **Simulation** dropdown menu, select the simulation for which you would like to view project benefits results.
3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.
4. Select the alternative(s) for which you would like to view impact response results and press **OK**.
5. The **Project Benefits Report** dialog box will appear (see **Figure 12.15**, page 12-15), by default on the **Urban Damages** report.
6. Select the **Structures** tab.
7. This shows the total damage incurred in your simulation with and without the projects (reservoirs and levees) you have created. Under the **Damage Categories** column, the damage categories you created in Chapter 4, Section 4.4 will appear.
8. From the **View By** list, results can be viewed by boundary.

9. The report contains results for the selected alternative. Results for the with- and without-project damages are reported by impact area and damage categories. At the bottom, users can view the total damages with and without projects.
10. If the selected alternative included projects in its definition, then results will be reported in the next two areas of the report.
11. When finished viewing the report, click **Close**. The **Project Benefits Report** dialog box will close.

12.6.3   Population Impacted Project Benefits Report

To view **Population Impacted Project Benefits Report**:

1. From the **Menu Bar**, select **Results→Project Benefits Report** (**Figure 12.14**).
2. A dialog box will appear. From the **Simulation** dropdown menu, select the simulation for which you would like to view project benefits results.
3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.
4. Select the alternative(s) for which you would like to view impact response results and press **OK**.
5. The **Project Benefits Report** dialog box will appear (see **Figure 12.15**, page 12-15), by default on the **Urban Damages** report.
6. Select the **Population** tab.
7. This shows the total damage incurred in your simulation with and without the projects (reservoirs and levees) you have created. Under the **Damage Categories** column, the damage categories you created in Chapter 4, Section 4.4 will appear.
8. From the **View By** list, results can be viewed by boundary.
9. The report contains results for the selected alternative. Results for the with- and without-project damages are reported by impact area and damage categories. At the bottom, users can view the total damages with and without projects.
10. If the selected alternative included projects in its definition, then results will be reported in the next two areas of the report.
11. When finished viewing the report, click **Close**. The **Project Benefits Report** dialog box will close.

12.6.4   Agricultural Damage Project Benefits Report

1. From the **Menu Bar**, select **Results→Project Benefits Report** (**Figure 12.14**).
2. A dialog box will appear. From the **Simulation** dropdown menu, select the simulation for which you would like to view project benefits results.

3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.
4. Select the alternative(s) for which you would like to view impact response results and press **OK**.
5. The **Project Benefits Report** dialog box will appear (see **Figure 12.15**, page 12-15), by default on the **Urban Damages** report.
6. Select the **Agriculture** tab.
7. This shows the total damage incurred in your simulation with and without the projects (reservoirs and levees) you have created. Under the **Damage Categories** column, the damage categories you created in Chapter 4, Section 4.4 will appear.
8. From the **View By** list, results can be viewed by boundary.
9. The report contains results for the selected alternative. Results for the with- and without-project damages are reported by impact area and damage categories. At the bottom, users can view the total damages with and without projects.
10. If the selected alternative included projects in its definition, then results will be reported in the next two areas of the report.
11. When finished viewing the report, click **Close**. The **Project Benefits Report** dialog box will close.

12.7 Annual Benefits Results

FIA has the ability to summarize annual damage/impacts and accomplishments by events. Reports available display damage/impacts and benefits for with and without projects side by side for events throughout the year and then total the accumulated damage/impacts and benefits for the year. This report should help Corps offices more efficiently develop the annual project benefit accomplishment reports that are required for submittal to Congress.

12.7.1 Build Annual Benefits Report

Before you view the **Annual Benefits Report**, you must build the report from simulations and alternatives that you select.

To build an **Annual Project Benefits Report**:

1. From the **Menu Bar**, select **Results→Annual Benefits Report→ Annual Project Benefits** (**Figure 12.16**).
2. The **Build Annual Benefits** dialog box will appear (**Figure 12.17**). A folder will appear for each computed simulation in your study. Double-click on each folder and check the boxes next to the alternatives you wish to include in the report.
3. At the top of the dialog box, enter a name for the report and a short description.
4. At the bottom of the dialog box, click **OK**.
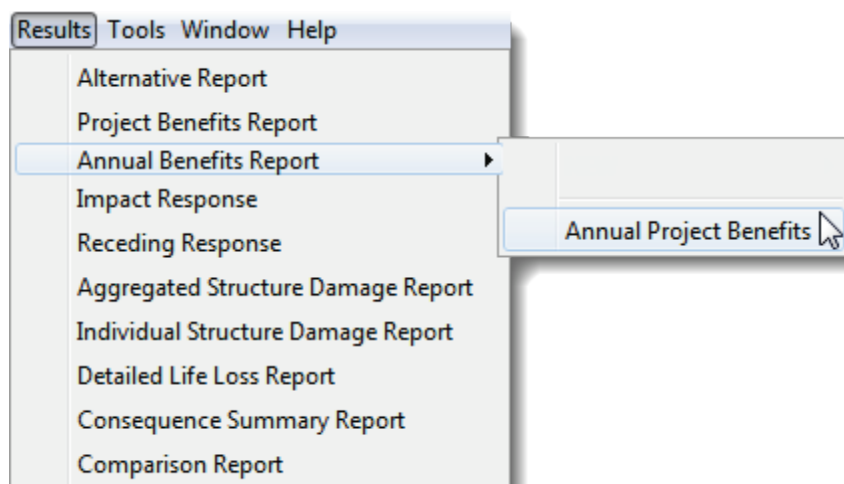
Figure 12.16 Results menu, Annual Project Benefits Report (Menu Bar).

5. The name of your new report will now appear under the **Results→Annual Benefits Report** menu (**Figure 12.18**).

12.7.2 View Annual Project Benefits Report

To view the **Annual Project Benefits Report**:

1. From the **Menu Bar**, select the **Results** menu. Point to **Annual Benefits Report**, point to the name of the report, and then click on the format in which you want to review the report (**Figure 12.19**).
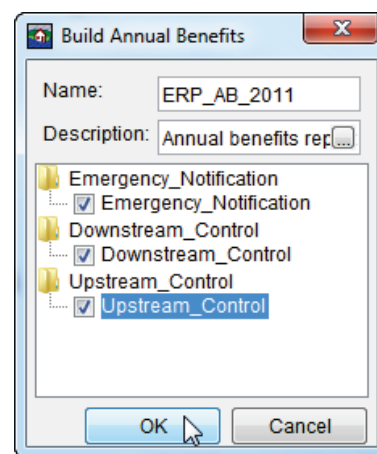
Figure 12.17 Build Annual Benefits dialog box.
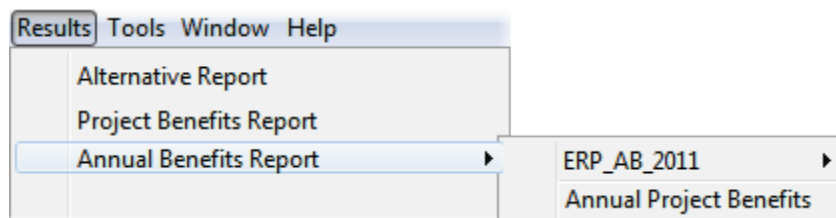
Figure 12.18 Completed annual project benefits report (Menu Bar).

2. You can view the report by impact area, state, or subbasin. You can also view your projects by state or subbasin.

3. Once you have decided how to view the report, the **Annual Project Benefits Report** dialog box will open (**Figure 12.20**).
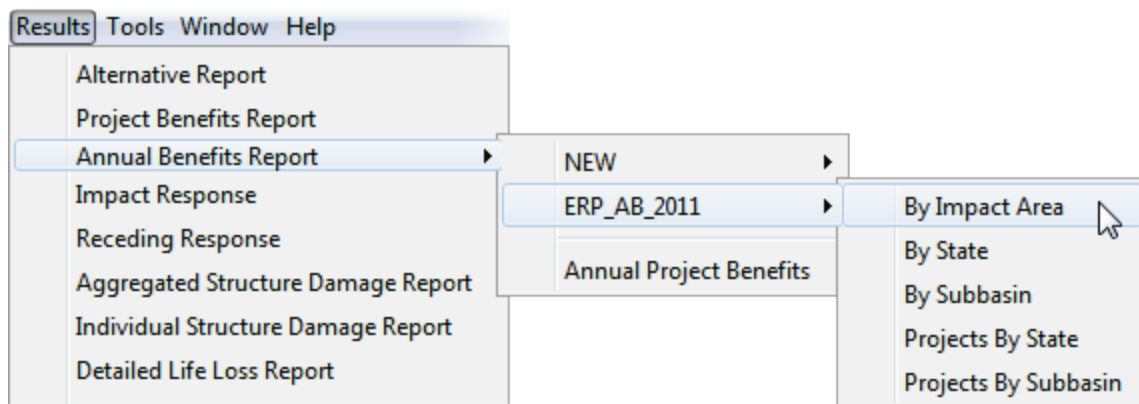
Figure 12.19 Viewing Annual Project Benefits reports by Impact Area

Figure 12.20 Annual Project Benefits Report

4. The report in **Figure 12.20** displays the information for one of the events that was selected for the annual benefits report - *JanFeb2002_Event*. Also, displayed is the alternative selected - *WithLevee*. There is a section like this for each simulation that was selected for this report.

5. Results for the with- and without-project damages are reported by impact area. For *Lock Haven* the **With Project** damage was *zero*, while the **Without Project** damage was *$26,572,000*.

8. The **Damage Prevented** area is reporting the damage prevented by project. So in **Figure 12.20**, the levee prevented *19,780,000* dollars worth of damage, while *Sayers* prevented *6,792,000* dollars worth of damage.

9. When finished viewing the report, click **Close**. The **Project Benefits Report** dialog box will close.

12.8   Impact Response Report

In simulations created using HEC-DSS hydrographs (time-series data), FIA continuously simulates conditions and evaluates damage during a flood event. As the program steps through a simulation, evaluating damage, it also searches through the rules in the **Impact Table** (entered in the **Impact Area Editor**; see Chapter 9, Section 9.7.9) to identify the stage thresholds that may be exceeded. When it finds a threshold that is exceeded, it records the time and date. At the conclusion of the simulation, FIA will display the results in the **Impact Response Report**. This report summarizes, for each impact area, thresholds that were exceeded, along with the time and date of the exceedance. The columns of the report represent the various simulation alternatives that have been evaluated. Also, from this report you can determine how various simulation alternatives would affect an impact area during a flood event.

To view the **Impact Response Report**:

1. From the **Menu Bar**, select **Results→Impact Response**.
2. A dialog box will appear (**Figure 12.21**). From the **Simulation** dropdown menu, select the simulation for which you would like to view impact response results.
3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.
4. Select the alternative(s) for which you would like to view impact response results and press **OK** (**Figure 12.21**).
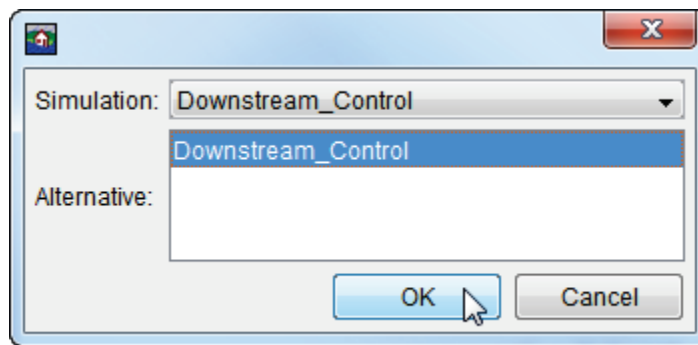
Figure 12.21   Simulation/Alternative selection dialog box.

5. The **Impact Response** dialog box will appear (see **Figure 12.22** below). This shows the *if-then* conditions you created in the **Impact**

Figure 12.22 The Impact Response Report

> **Table** of the **Impact Area Editor** (see Chapter 9, Section 9.9.9) for each stage and the times at which they were exceeded. It will also show the maximum flood stage for each impact area.

7. Click **Close**, the **Impact Response Report** dialog box will close.

12.9   Receding Response Report

Using the hydrographs (time-series data) defined for the impact area, FIA continuously simulates conditions and evaluates potential environmental and operational problems due to low flow. As the program steps through a simulation, evaluating damage, it also searches through the rules in the **Receding Table** (entered in the **Impact Area Editor**, Chapter 9, Section 9.7.10) to identify the stage thresholds that need to be maintained or exceeded. When it finds a threshold that is not met, it records the time and date. At the conclusion of the simulation, FIA will display the results in the **Receding Response Report**. This report summarizes, for each impact area, thresholds that were not met, along with the time and date of the threshold not being met. The columns of the report represent the various simulation alternatives that have been evaluated. Also, from this report you can determine how various simulation alternatives would affect an impact area during a low flow event.

To view the **Receding Response Report**:

1. From the **Menu Bar**, select **Results➤Receding Response**.
2. A dialog box will appear. From the **Simulation** dropdown menu, select the simulation for which you would like to view impact response results.
3. Once a simulation is selected, the alternatives associated with that simulation will appear in the **Alternative** box.

4. Select the alternative(s) for which you would like to view impact response results and press **OK** (**Figure 12.21**).

5. The **Receding Response Report** dialog box will appear (see **Figure 12.22** above). This shows the *if-then* conditions you created in the **Recession Table** of the **Impact Area Editor** (see Chapter 9, Section 9.7.10) for each stage and the times at which they were exceeded. It will also show the maximum flood stage for each impact area.

Figure 12.23 The Receding Response Report

7. Click **Close**, the **Receding Response Report** dialog box will close

12.10   Printing and Exporting Reports

The Project Benefits, Annual Project Benefits, Impact Response, and Recession Response reports share the same method for printing and saving. From each report, there is a shortcut menu that allows you to print the report, preview the report before you print, and save the report to an ASCII or text (.txt) file (tab delimited). Right-clicking on the active report accesses the shortcut menu. Another way to print or export the reports is by clicking on **Print** or **Save to File**. Either method accesses the same dialog boxes.

12.10.1 Printing Reports

The output will be a print of the screen image of the report you actively have open. If items in the columns of the report are not printing, you can use the mouse to resize the column widths and the width of the entire report dialog box if you need to see more detail. To do so, roll the mouse pointer over one of the lines that delimit the columns or over the dialog box border. You can click on that line with the left mouse button and drag the mouse from side to side to indicate how wide you want the columns or dialog box. If you end up highlighting some cells in the table, then you probably moved the mouse slightly just before you pressed the button.

To print a report:

1. Open the report you wish to print. To open the Project Benefits, Annual Project Benefits, Impact Response, or Recession Response Reports, refer to the previous sections 12.6, 12.7, 0, and 12.9, respectively.
2. At the bottom of the report dialog, click **Print**. The **Print Properties** dialog box (see **Figure 12.4**, page 12-3). You could also right-click on the report and select **Print** from the shortcut menu.
4. Make the same adjustments for printing as described in Section 12.1.2 (page 12-2).
5. After you have everything set, click **Print**. Your default system print settings dialog box will appear. Select the printer, and any other options you wish to set, click **OK**. The report will then be printed on the selected printer.

12.10.2   Print Preview

You can also preview the report before printing.

To view a print preview:

1. Open the report you wish to preview. To open the Project Benefits, Annual Project Benefits, Impact Response, or Recession Response Reports, refer to the previous sections 12.6, 12.7, 12.8, and 12.9, respectively.
2. Right-click anywhere one the report. From the shortcut menu, click **Print Preview**.
3. The **Print Properties** dialog box will appear (see **Figure 12.4**, page 12-3).
4. Make the same adjustments for printing as described in Section 12.1.2 (page 12-2).
5. After you have everything set, click **Preview**. The **Print Preview** dialog box will appear (see **Figure 12.12**, page 12-12).
6. If you like what you see in the **Print Preview** dialog box, you can print by clicking **Print**, which opens the system **Print** dialog box
7. Select the printer, and any other options you wish to set, click **OK**. The report will then be printed on the selected printer.

12.10.3   Exporting Reports to File

You can also export the reports to an ASCII delimited file.

To export results:

1. Open the report you wish to export. To open the Project Benefits, Annual Project Benefits, Impact Response, or Recession Response

Reports, refer to the previous sections 12.6, 12.7, 12.8, and 12.9, respectively.

3. At the bottom of the report dialog box, click **Save to File**. The **Table Export Options** dialog box will open. You could also right-click on the report, from the shortcut menu, click **Export**, and the same dialog box will open.

4. You can specify the delimiter to be used, the default is **TAB**. Your choices are **TAB, SPACE, COMMA**, or **COLON**.

6. You can also include the column headers of the report (default), include gridlines, and include the title of the report.

7. Once you are set, click **OK**, the system **Save Browser** will open.

8. Select where you want the file to be saved and a filename. If you want the file to have an extension, you must enter your own; there is no default extension.

9. Click **Save**, the **Save Browser** will close. The filename that you entered will now be saved on your computer.

Appendix A

References

Hydrologic Engineering Center, 2006b. *HEC-DSSVue, HEC Data Storage System Visual Utility Engine User's Manual, Version 1.2, CPD-79*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 2009. *HEC-RAS, River Analysis System User's Manual, Version 4.0.1, CPD-68*, U.S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 1995. *HEC-PBA Project Benefit Accomplishment Package User's Manual, CPD-67*, U.S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 2007. *HEC-ResSim, Reservoir System Simulation User's Manual, Version 3.0,* U.S. Army Corps of Engineers, Davis, CA.

Rogers, G. O. and J. H. Sorensen, 1991. "Diffusion of Emergency Warning: Comparing Empirical and Simulation Results," in *Risk Analysis Prospects and Opportunities*, C. Zerros et al. (eds.), Plenum Press, New York, p. 117 - 134.

Appendix B

Data Preparation

The geographic information system (GIS) pre-processing that is required for an FIA study is handled through the use of the ESRI ArcGIS Desktop software. Information and availability about the ArcGIS Desktop software is available from ESRI ([www.esri.com)](www.esri.com). The software should include ArcMap, the ArcGIS 3D Analyst extension, and the ArcGIS Spatial Analyst extension.

Besides developing the layers that HEC-FIA requires, the GIS pre-processing steps outlined in this chapter will ensure that all geographic data layers are in the same projected coordinate system.

There are several GIS pre-processing steps that need to be completed before creating a study in HEC-FIA. Depending on the level of detail in the study, not all of the following will have to be utilized:

1. Preparation of maximum inundation depth grids
2. Preparation of duration and arrival time grids
3. Creation of maximum inundation area polygon
4. Generation of the hazard area boundary
5. Preparation of a digital terrain model
6. Preparation of arrival time grid for depths two feet and greater
7. Preparation of FEMA (US Department of Homeland Security, Federal Emergency Management Agency) HAZUS (HAZards United States) data
8. Preparation of various items developed by HEC-GeoRAS (Hydrologic Engineering Center's ArcGIS extension for processing geospatial data for HEC-RAS)
9. Preparation of impact area polygons
10. Preparation of double warning time polygon
11. Agricultural grid preparation.

Each of these steps is explained in the following sections.

B.1   Preparing Maximum Inundation Depth Grids

For rapid consequence assessment, flood inundation information for a specific flood scenario is supplied to HEC-FIA in the form of depth grids. Depth grids used for the assessment are typically generated by a hydraulic engineer using HEC-GeoRAS, RAS Mapper or another hydraulic model.

The maximum inundated depth grid is required for each flood scenario to be analyzed. Note, if the maximum inundation depth grid was produced in RAS Mapper then the depth grid is already in the binary format (.flt) required by HEC-FIA. The RAS Mapper maximum inundation depth grid is located in the "Max WS" folder of the RAS Mapper results directory.

To convert the depth grid(s) to binary format (.flt):

1. Start **ArcCatalog**, the **ArcCatalog** window will open (Figure B.1).

Figure B.1 ArcCatalog Window.

2. If the **ArcToolbox Pane** is not present, click **ArcToolbox** on the **Standard Toolbar** (Figure B.2) and the **ArcToolbox Pane** will display in the **ArcCatalog** window (Figure B.1).

Figure B.2 ArcCatalog Window—Standard Toolbar.

3. From the **ArcToolbox Tree**, double-click **Conversion Tools**, double-click **From Raster**, and then double-click **Raster to Float** (Figure B.3).

4. The **Raster to Float** dialog box will open (Figure B.4). Click the 📂 next to the **Input raster** box to select a grid.

5. The **Input raster** browser will open (Figure B.5). Browse to the location of the depth grid file select it. Click **Add**, the **Input raster** browser will close, and the path and the name of the selected depth grid file will be displayed in the **Input raster** box on the **Raster to Float** dialog box (Figure B.4).

Figure B.3 ArcToolbox Tree.
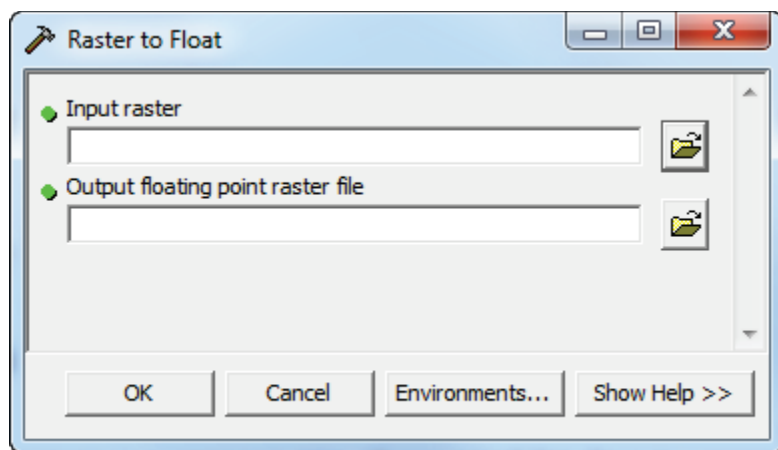
Figure B.4 Raster to Float dialog box.
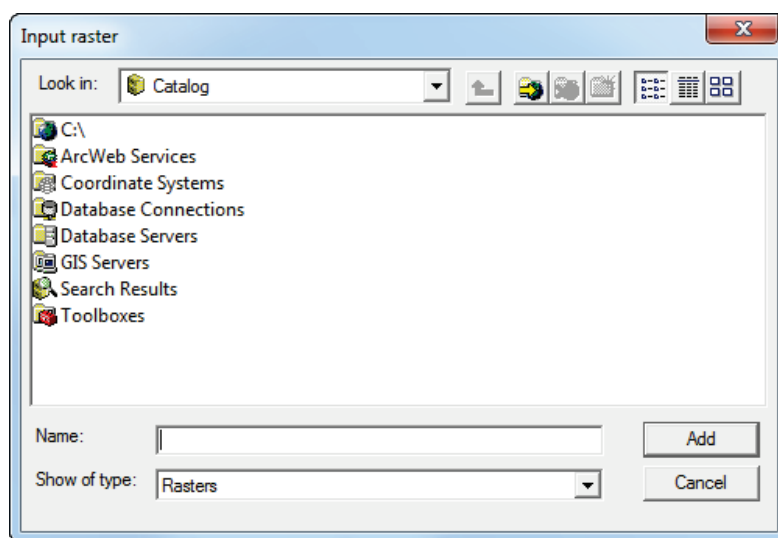
Figure B.5 Input raster browser.

6. By default the output floating raster file will be stored to the same directory as the selected depth grid file. In the **Output floating point raster file** box (Figure B.6) the path and the default name for the binary file will appear. The default name is the depth grid filename with *RasterT_* appended to the front, and appended to the end if *1.FLT* (i.e., *RasterT_dp001.FLT*). Modify this to represent flood scenario represented by the depth grid (i.e. *FP_Fail.flt*).
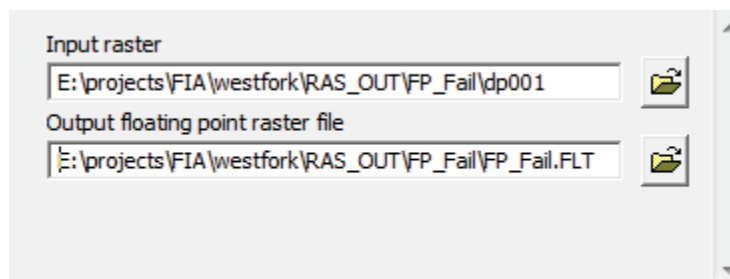
Figure B.6 Output floating point raster file default name (Raster to Float dialog box).

7. Click **OK**. The **Raster to Float** dialog box will close and the **Raster to Float Compute** dialog box will open (Figure B.7). This dialog box notifies the user of the status of the compute and how long it took to create the binary file. When finished viewing information, click **Close** to close the **Raster to Float Compute** dialog box.
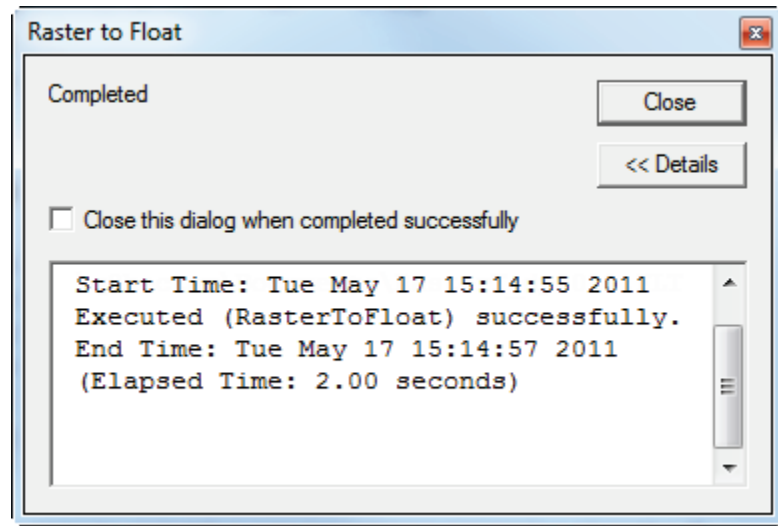
Figure B.7 Raster to Float compute dialog box.

8. Now in the selected directory, there are two files, *.flt and *.hdr. The depth grid has been successfully converted to a binary format.

B.2   Preparing Duration and Arrival Time Grids

This step is only required if using duration and arrival time grids in place of hydrograph data. Duration grids are only required for agricultural damages. If agricultural damage is not part of the assessment then duration grids do not need to be prepared. Arrival time grids are required for both agricultural and life loss calculations.

Duration grids and arrival time grids are generally produced from either ArcGIS plug-ins or RAS Mapper. The duration grid is defined as the duration of time a grid cell is wet. The arrival time grid contains the amount of time it takes for water stage to reach a predefined threshold at each cell. Typically the arrival time threshold will be two (2) feet. If life loss calculations are to be computed with arrival time grids, make sure the threshold for arrival time (default of 2 feet) is the same threshold used to define the hazard boundary for each scenario.

For an agricultural damage assessment, the arrival time threshold will typically be zero (0) feet.

B.2.1   Duration Grid Preparation

To convert the duration grid to binary format (.flt):

1. Start **ArcCatalog**. The **ArcCatalog** window will open (see Figure B.1, page B-2).
2. If the **ArcToolbox Pane** is not present, click **ArcToolbox** on the **Standard Toolbar** (see Figure B.2, page B-2) and the **ArcToolbox Pane** will display in the **ArcCatalog** window.
3. From the **ArcToolbox Tree**, double-click **Conversion Tools**, double-click **From Raster**, and then double-click **Raster to Float** (see Figure B.3, page B-2).
4. The **Raster to Float** dialog box will open (see Figure B.4). Browse to the location of the duration grid file by clicking next to the **Input raster** box.
5. The **Input raster** browser will open (see Figure B.5). Browse to the location of the duration grid file, click on the duration grid filename. Click **Add**, the **Input raster** browser will close (see Figure B.5), and the path and the name of the selected duration grid file (i.e., *E:\projects\FIA\westfork\Duration\FP_Fail \FP_F_Duration*) will appear in the **Input raster** box on the **Raster to Float** dialog box (see Figure B.4).
6. By default the software will create the output floating point raster file to the same directory as the selected duration grid file. In the **Output floating point raster file** box (see Figure B.6) the path and the default name for the binary file (i.e., *E:\projects\FIA\westfork\Duration\FP_Fail\RasterT_FP_F1.FLT*) will appear. Modify the float file name to represent the flood scenario represented by the duration grid (i.e. *FP_Fail_Duration.FLT*).
7. Click **OK**, the **Raster to Float** dialog box will close. The **Raster to Float Compute** dialog box will open (see Figure B.7). This dialog box notifies the user of the status of the compute and how long it took to create the binary file. When finished viewing information, click **Close**. The **Raster to Float Compute** dialog box will close.
8. Now in the selected directory, there are two files, *.flt and *.hdr. The duration grid has been successfully converted to a binary format.

B.2.2   Arrival Time Grid Preparation

To convert the arrival time grid to binary format (.flt):

1. Start **ArcCatalog**. The **ArcCatalog** window will open (see Figure B.1).
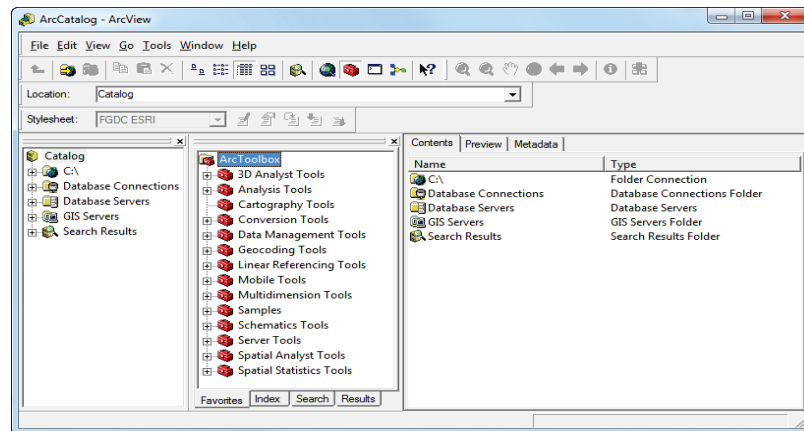
2. If the **ArcToolbox Pane** is not present, click **ArcToolbox** on the **Standard Toolbar** (see Figure B.2) and the **ArcToolbox Pane** will display in the **ArcCatalog** window.

3. From the **ArcToolbox Tree**, double-click **Conversion Tools**, double-click **From Raster**, and then double-click **Raster to Float** (see Figure B.3).

4. The **Raster to Float** dialog box will open (see Figure B.4). Browse to the location of the arrival time grid file by clicking 📂 next to the **Input raster** box.

5. The **Input raster** browser will open (see Figure B.5). Browse to the location of the arrival time grid file, click on the grid filename. Click **Add**, the **Input raster** browser will close, and the path and the name of the selected arrival time grid file will appear in the **Input raster** box on the **Raster to Float** dialog box.
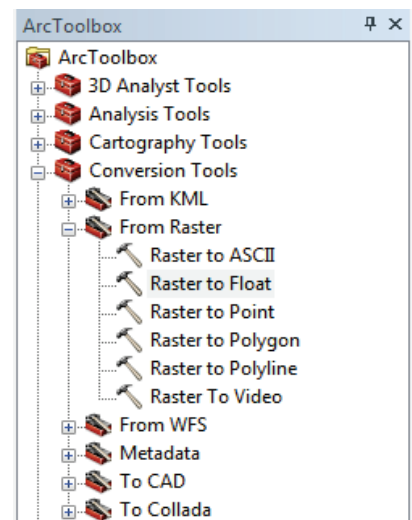
6. By default the software will create the output floating point raster file to the same directory as the selected arrival time grid file. In the **Output floating point raster file** box (see Figure B.6) the path and the default name for the binary file (i.e., *E:\projects\FIA\westfork\ArrivalTime\FP_Fail\RasterT_FP_F1.FLT*) will appear. Modify the float file name to represent the flood scenario represented by the arrival time grid (i.e. *FP_Fail_Arrival.FLT*).

7. Click **OK**. The **Raster to Float** dialog box will close and the **Raster to Float Compute** dialog box will open (see Figure B.7). This dialog box notifies the user of the status of the compute and how long it took to create the binary file. When finished viewing information, click **Close**. The **Raster to Float Compute** dialog box will close.

8. Now in the selected directory, there are two files, *.flt and *.hdr. The arrival time grid has been successfully converted to a binary format.

B.3 Maximum Inundated Area Polygon

A polygon that represents the maximum inundated area for the largest flood scenario is necessary to trim other required data down to a manageable size. The inundated area polygon for the maximum flood inundation scenario is typically generated by the hydraulic engineer while performing inundation mapping in HEC-GeoRAS (bounding polygons). These bounding polygons are stored in the HEC-GeoRAS geodatabase for a given study. This maximum inundated area will need to be converted to a shapefile for later use.

To convert the maximum inundated area bounding polygon to a shapefile:

1. Start **ArcCatalog**. The **ArcCatalog** window will open (see Figure B.1).

2. From the first pane (the **Catalog Tree),** browse to the folder that contains the geodatabase file for your study.

3. Once the geodatabase file appears in the **Contents** tab of the main window (Figure B.8), double-click on the geodatabase filename (i.e., *Turman_SDF.mdb*). Next, double-click on the geodatabase feature dataset (i.e., *RasResults*).
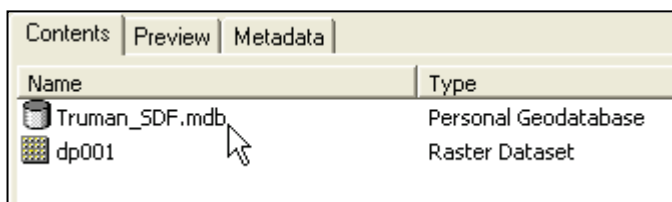
Figure B.8 Contents Tab—Geodatabase File.

4. The bounding polygon file created by HEC-GeoRAS should appear in the list (i.e., *bP001*). Right-click on the bounding polygon filename (**Figure B.9**). From the shortcut menu, point to **Export** and select **To Shapefile (single)**.
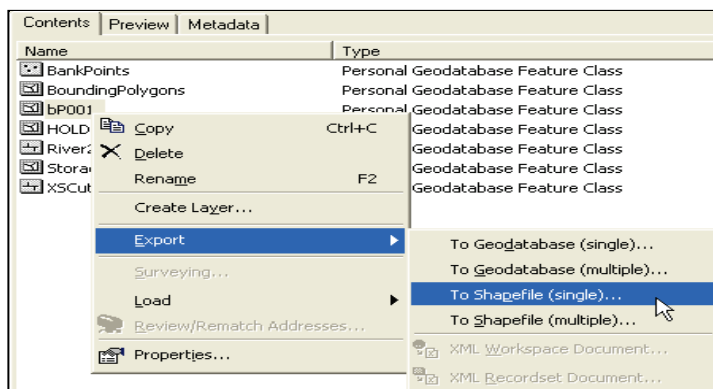
Figure B.9 Export Shortcut Menu.

5. The **Feature Class to Feature Class** dialog box will open (**Figure B.10**).
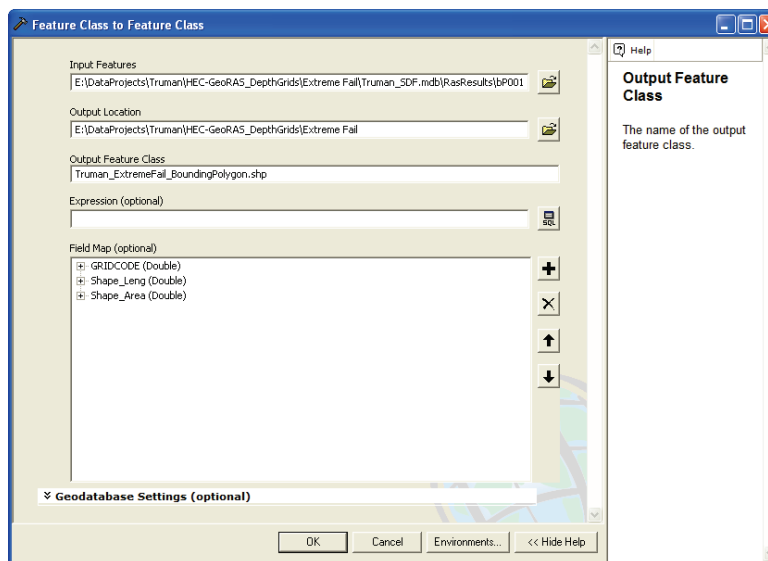
Figure B.10 Feature Class to Feature Class dialog box.

6. The **Input Features** box should contain the name of the selected bounding polygon file. Choose the location to store the shapefile by clicking ![folder icon] next to the **Output Location** box. The **Output Location** browser will open (Figure B.11). Browse to the folder where you want to store the shapefile, click **Add**. The **Output Location** browser will close, and the pathname with the selected folder will appear in the **Output Location** box of the **Feature Class to Feature Class** dialog box.
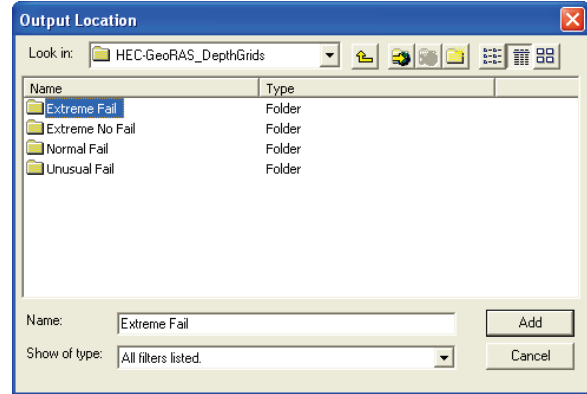
Figure B.11 Output Location Browser.

7. In the **Output Feature Class** box, provide a name for the shapefile (i.e., *Truman_ExtremeFail_BoundingPolygon*)

8. Click **OK.** The **Feature Class to Feature Class** dialog box will close, and the **Feature Class to Feature Class Compute** dialog box will open (Figure B.12). This dialog box will contain information on if creation of the float file was successful and how long it took to create the binary file. Click **Close**. The **Feature Class to Feature Class Compute** dialog box will close and the shapefile will appear in the location you selected.
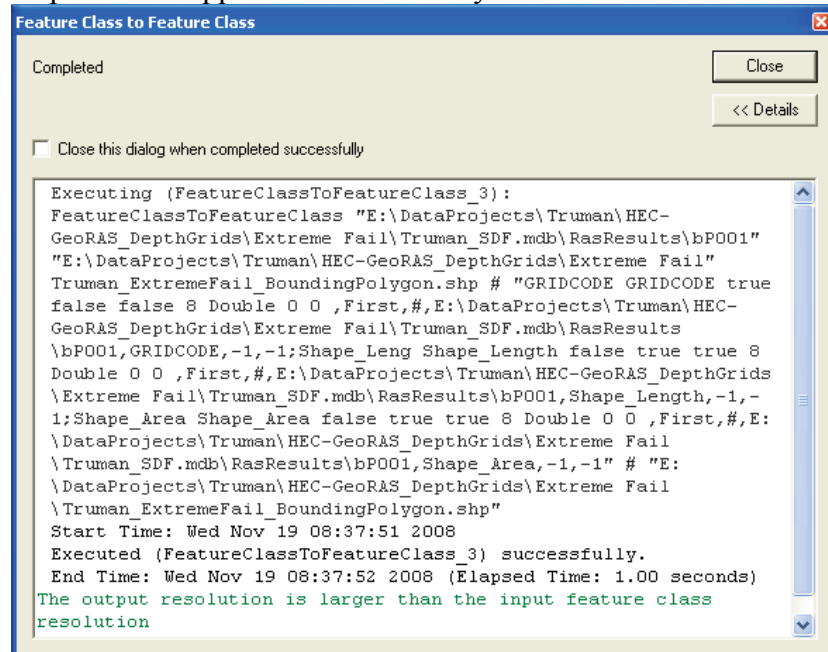
Figure B.12 Feature Class to Feature Class Compute dialog box.

B.4   Generate Hazard Area Boundary

During the evacuation portion of the life loss computation in HEC-FIA, the time it takes for people in a given structure to evacuate safely (defined as the time from leaving a structure to reaching a safe location) can be defined by either:

- Entering the exact time (minutes) it takes for the inhabitants of each structure to reach a safe location, or

- Providing a hazard area boundary and nominal evacuation velocity (mph) and letting HEC-FIA compute the time required for the inhabitants of each structure to evacuate to safety.

Typically, a hazard area boundary and nominal velocity will be entered in HEC-FIA to estimate the evacuation times for a given event. The hazard area boundary should be determined by creating a polygon that represents the extents of flooding where depth is greater than two feet. Each flood event being simulated will have a separate hazard area boundary. To create a hazard area boundary, follow the steps in the following sections.

B.4.1   Reclassify Depth Grid Cells

Reclassify depth grid cells for the maximum depth grid to values of one (1) if greater than two feet deep and two (2) if less than or equal to two feet deep:

1.   Start **ArcMap**. The **ArcMap** window will open (Figure B.13). By default, a **new empty map** is selected, click **OK**.
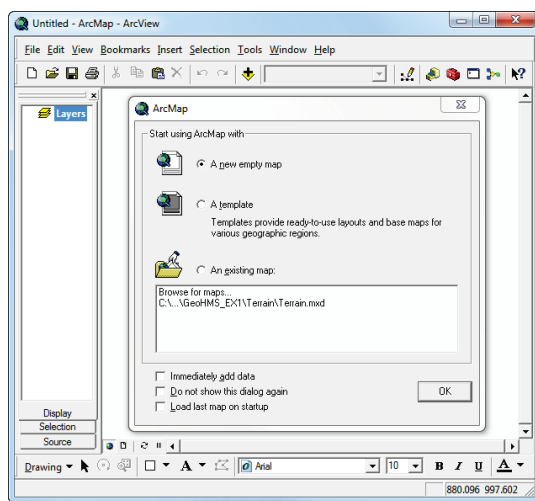
Figure B.13 ArcMap Window.

2.   If the **Spatial Analyst Toolbar** is not available from the **ArcMap** window, right-click on the empty area next to the **Menu Bar** (Figure B.14). From the shortcut menu, click **Spatial Analyst**, the
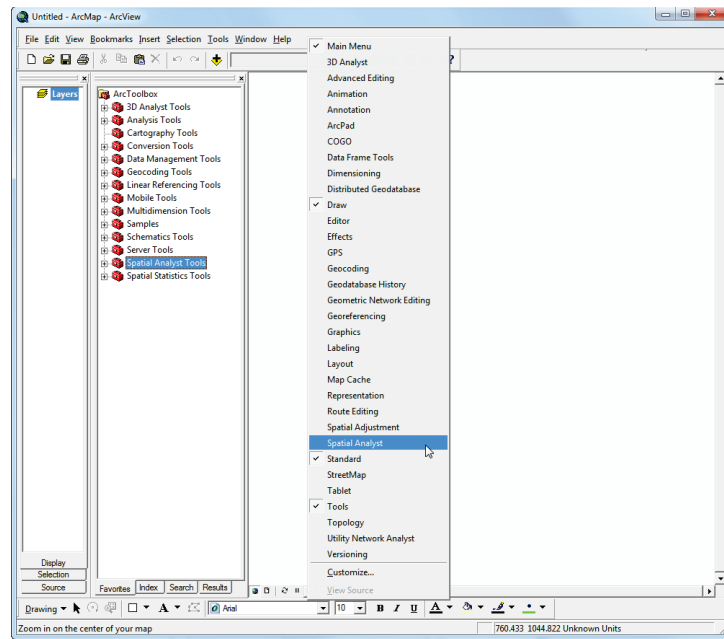
Figure B.14 Toolbar Setup shortcut menu.

Spatial Analyst Toolbar (Figure B.15) will appear in the **ArcMap** window in the toolbar area.
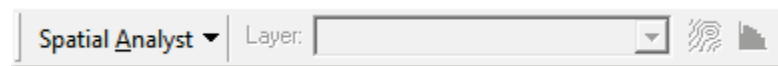
Figure B.15 Spatial Analyst Toolbar.

3.   From the **Spatial Analyst Toolbar**, click **Spatial Analysis** and then select **Reclassify** from the shortcut menu. The **Reclassify** dialog box will open (Figure B.16).
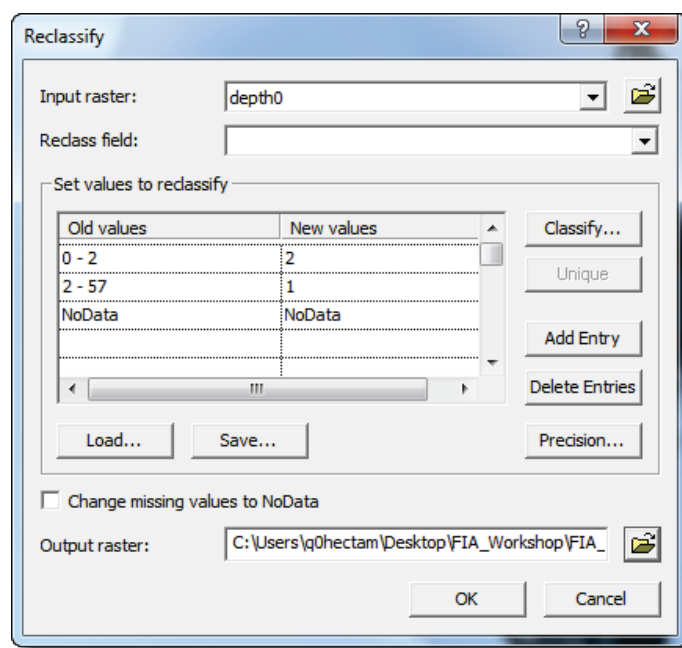
Figure B.16 Reclassify dialog box.

4.   Input raster is the maximum depth grid (*depth0*) for the flood scenario being assessed. Select the depth grid file from the dropdown list or by clicking open ☞ next to the **Input raster** box.

5.   The **Choose an input raster** browser will open. Browse to the location of the depth grid file, click on the depth grid filename and then click **Add**. The **Choose an input raster** browser will close, and the filename of the grid will appear in the **Input raster** box on the **Reclassify** dialog box.

6.   Leave the **Reclass field** box blank.

7.   Default values will appear in the **Set values to reclassify** box (Figure 3.18). These values (both old and new) need to be reset. Set all values less than or equal to two (2) feet to a new value of two (2) and values from two (2) feet up to the highest value in the depth grid to a new value of one (1). For an example, see Figure 3.16 (page 3-8). Note that when entering the old value range a space is required on both sides of the dash (enter 0 - 2, not 0-2).
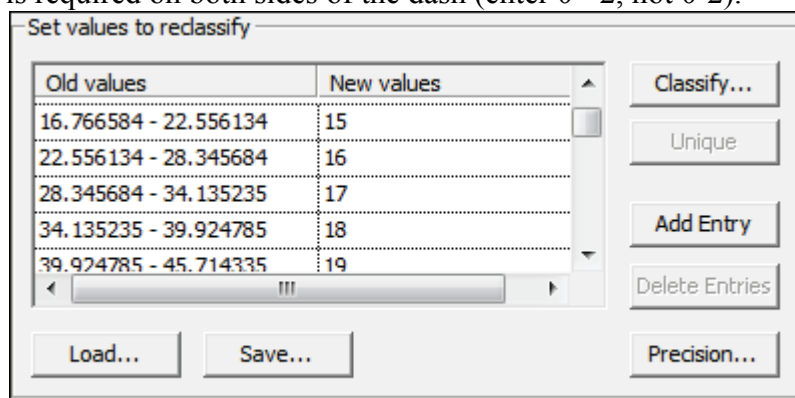
Figure B.17 Set Values to Reclassify (Reclassify dialog box).

8.   In the **Output raster** box, type the desired pathname for the depth grid or click open ☞ to browse to the place you would like to save the grid.

9.   Click **OK**. The hazard area grid will be created. When that is finished, the **Reclassify** dialog box will close and the hazard area boundary grid will display in the ArcMap window.

B.4.2   Generate Polygon Shapefile of the Hazard Area Boundary

Generate a polygon shapefile that represents the hazard area boundary (can be done in ArcMap or ArcCatalog):

1.   If the **ArcToolbox Pane** is not present, click **ArcToolbox** 🔴 on the **Standard Toolbar** (see Figure 3.1, page 3-1) and the **ArcToolbox Pane** will display in the ArcMap or ArcCatalog window.

2. From the **ArcToolbox Tree**, double-click **Conversion Tools**, double-click **From Raster**, and then double-click **Raster to Polygon** (see Figure B.3). The **Raster to Polygon** dialog box will open (Figure B.18).
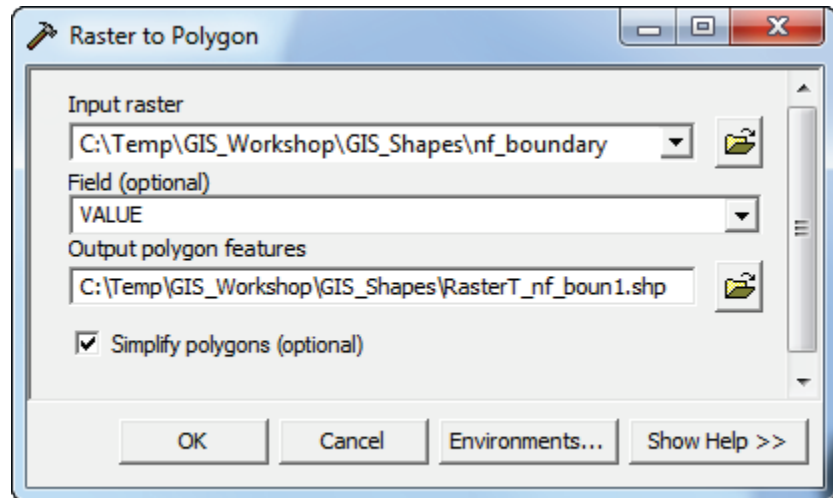
Figure B.18 Raster to Polygon dialog box.

3. The input raster file is the hazard area boundary grid that was created in the previous section. Select the location of the hazard area boundary grid by clicking open ☞ next to the **Input raster** box (Figure B.18). Browse to the location of the hazard area boundary grid, and select it. Click **Add**, the **Input raster** browser will close, and the path and the name of the selected hazard area boundary file will appear in the **Input raster** box on the **Raster to Polygon** dialog box.

4. The **Field** box should be set to **VALUE**; this is used to assign values from the cells in the hazard area boundary file to the polygons in the shapefile.

5. By default the software will store the output shapefile to the same directory as the selected hazard area boundary grid. In the **Output polygon features** box (see Figure 3.20, page 3-10), the path and the default name for the shapefile will appear. Change this name to symbolize the depth grid it represents (i.e. *normal_fail_hazardarea.shp*)

6. Select **Simplify polygons** so that the software will smooth the polygons in ☞ simpler shapes.

7. Click **OK**. The **Raster to Polygon** dialog box will close and the **Raster to Polygon Compute** dialog will open. This dialog box notifies the user of the status of the compute and how long it took to create the binary file. When finished viewing information, click **Close**. The **Raster to Polygon Compute** dialog box will close.

B.4.3   Editing the Hazard Area Boundary Shapefile

All polygons with a value of one (1) in the output polygon dataset represent the hazard area boundary. Edit the polygon shapefile and select all polygons with a value of two (2) and delete them.

To edit the hazard area boundary shapefile:

1. From the **Editor Toolbar** click **Editor** (Figure B.19), from the shortcut menu click **Start Editing**.
2. Right click on the shapefile that was created in the previous section. From the shortcut menu (Figure B.20), click **Open Attribute Table**. The attribute table for the hazard area polygon file will open (Figure B.21).
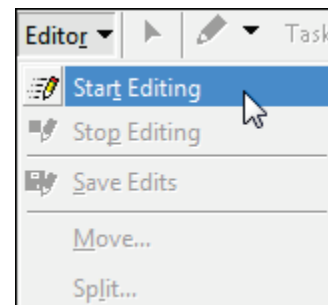
Figure B.19 Start Editing Menu Item.
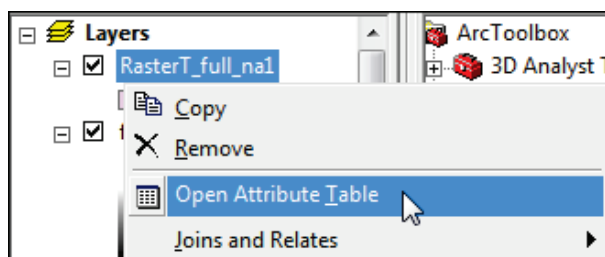
Figure B.20 Open Attribute Table Menu Item.

3. Click **Options** (Figure B.21). From the shortcut menu, click **Select by Attributes**. The **Select by Attributes** dialog box will open (Figure B.22).
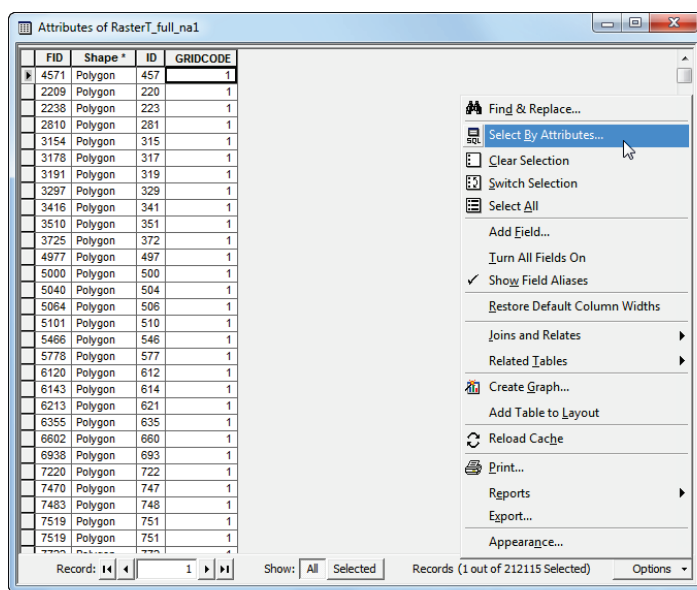
Figure B.21 Attribute Table.
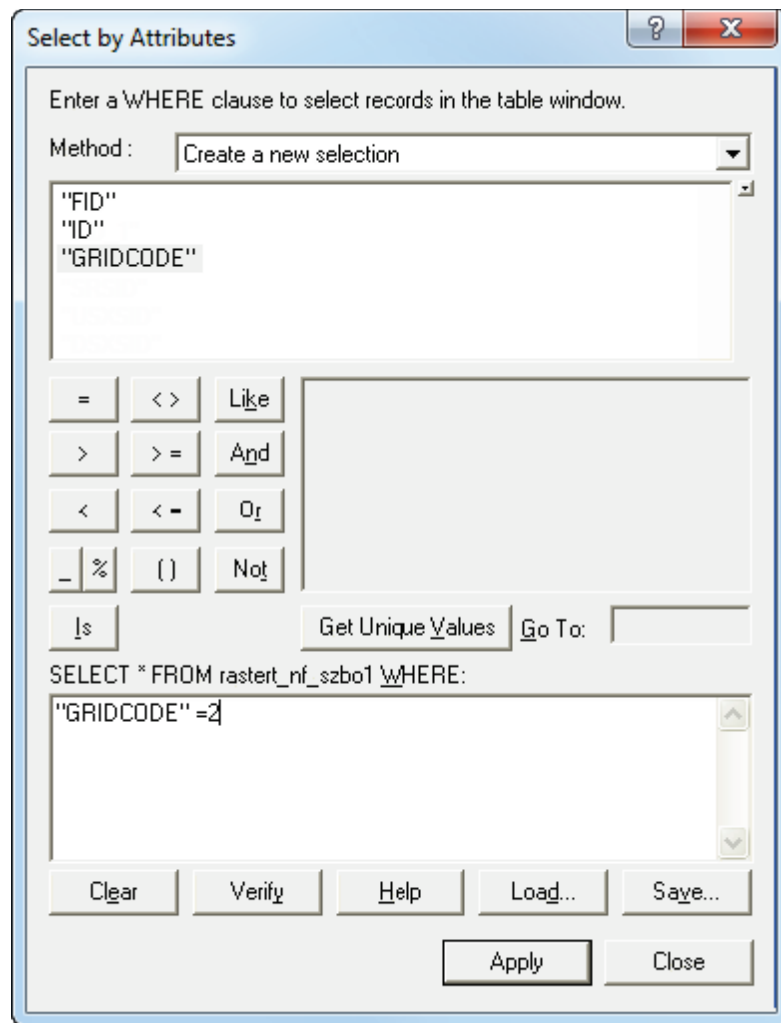
Figure B.22 Select by Attributes dialog box.

4. Double-click on **"GRIDCODE"**, in the **SELECT * FROM** *shapefile name* **WHERE** box "GRIDCODE" will appear. Click the equals sign button and then enter 2.

5. Click **Apply**, in the attribute table all the rows with a *GRIDCODE* equal to two (2) will be highlighted in a cyan color. If you have the shapefile displayed in the ArcMap window, the same items will be highlighted in cyan.

6. Click **Close**, the **Select by Attributes** dialog box will close.

7. From the attribute table dialog box push the **Delete** key on your keyboard. All of the highlighted rows will be deleted. Click ⊠ to close the attribute table.

8. From the **Editor Toolbar** click **Editor** (see Figure B.19). From the shortcut menu click **Save Edits**. Now the hazard area boundary shapefile has been edited to contain only the polygons that delineate areas where the maximum depth is greater than two feet.

9. Repeat all steps in this section for each depth grid in the study.

B.5   Preparing Digital Terrain Model

Terrain data must be in binary grid format for use in HEC-FIA. Terrain information is required to assign ground elevations to a structure inventory generated in HEC-FIA, to update ground elevations in an existing structure inventory, and to determine the arrival time at the hazard boundary during the evacuation stage of the life loss compute. To prepare a terrain grid for importing into HEC-FIA, follow the steps in this section.

B.5.1   Clip the Original Digital Terrain Grid

If the existing terrain grid extends far beyond the limits of the study, you can clip it the same area that is covered by the maximum inundated depth grid to make using it within HEC-FIA more efficient. If the terrain grid is of manageable size (less than 1GB in size), then skip to Section B.5.2.

To clip the digital terrain grid:

1. Start **ArcMap**. The **ArcMap** window will open (see Figure B.13).
2. If the **ArcToolbox Pane** is not present, click **ArcToolbox** 🔴 on the **Standard Toolbar** (see Figure B.1) and the **ArcToolbox Pane** will display in the **ArcMap** window.
3. From the **ArcToolbox Tree**, double-click **Spatial Analyst Tools**, double-click **Extraction**, and then double-click **Extraction by Mask** (Figure B.23).
4. The **Extract by Mask** dialog box will open (see Figure 3.27, page 3-14). Browse to the location of the digital terrain grid file by clicking open 📂 next to the **Input raster** box.
5. The **Input raster** browser will open (see Figure B.5). Browse to the location of the digital terrain grid file, click on the digital terrain gird filename. Click **Add**, the **Input raster** browser will close, and the pathname of the selected digital terrain grid file (i.e., *C:\Temp\GIS_Workshop\GIS_Shapes\terrain*) will appear in the **Input raster** box on the **Extract by Mask** dialog box (Figure B.24).
6. Browse to the location of the maximum inundated area bounding polygon shapefile (created in Section B.3) by clicking 📂 next to the **Input raster or feature mask data** box. The **Input raster or feature mask data** browser will open (Figure B.25).
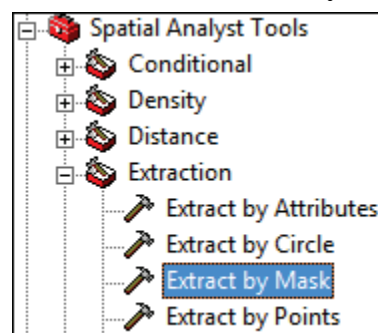
Figure B.23 Spatial Analyst Tools (ArcToolbox Tree).
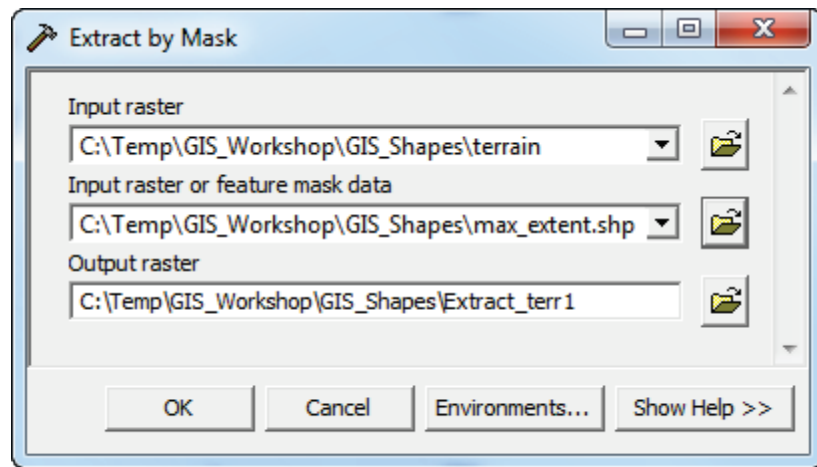
Figure B.24 Extract by Mask dialog box.
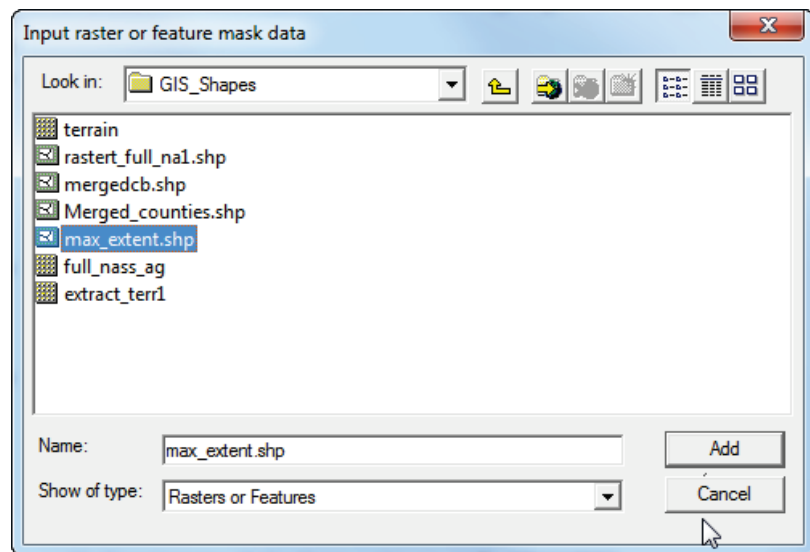
Figure B.25 Input raster or feature mask data browser.

7. Browse to the location of the maximum inundated area bounding polygon shapefile, click on the shapefile filename. Click **Add**, the **Input raster or feature mask data** dialog box will close, and the path and the name of the selected maximum inundated area bounding polygon shapefile will appear in the **Input raster or feature mask data** box on the **Extract by Mask** dialog box.
8. By default the software will create the masked terrain digital grid in the same location as the original terrain digital grid.
9. Click **OK**, the **Extract by Mask** dialog box will close and the **Extract by Mask Compute** dialog box will open (Figure B.26). This dialog box notifies the user of the status of the compute and how long it took to create the masked digital terrain grid file. When finished viewing information, click **Close**, the **Extract by Mask Compute** dialog box will close and the shapefile will appear in the location you selected.

Now in the selected directory, the masked terrain digital grid file will appear.

B.5.2   Convert Terrain Grid to a Binary Format

Convert the clipped digital terrain grid to binary format (.flt):
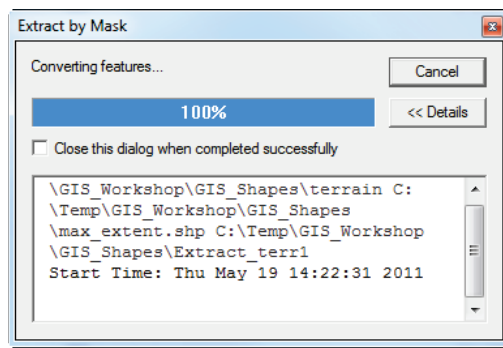
Figure B.26 Extract by Mask compute dialog box.

1. Start **ArcMap**. The **ArcMap** window will open (see Figure B.13).
2. If the **ArcToolbox Pane** is not present, click **ArcToolbox** on the **Standard Toolbar** (see Figure B.1) and the **ArcToolbox Pane** will display in the **ArcMap** window.
3. From the **ArcToolbox Tree**, double-click **Conversion Tools**, double-click **From Raster**, and then double-click **Raster to Float** (see Figure B.3).
4. The **Raster to Float** dialog box will open (see Figure B.4). Browse to the location of the clipped digital terrain grid file by clicking next to the **Input raster** box.
5. The **Input raster** browser will open (see Figure B.5). Browse to the location of the clipped digital terrain grid file, click on the clipped digital terrain grid filename. Click **Add**, the **Input raster** browser will close, and the pathname of the selected clipped digital terrain grid file will appear in the **Input raster** box on the **Raster to Float** dialog box.
6. By default the software will create the output floating point raster file to the same directory as the selected clipped digital terrain grid file. In the **Output floating point raster file** box (see Figure B.6) the path and the default name for the binary file will appear.
7. Click **OK**, the **Raster to Float** dialog box will close. The **Raster to Float Compute** dialog box will open (see Figure B.7). This dialog box notifies the user of the status of the compute and how long it took to create the binary file. When finished viewing information, click **Close**, the **Raster to Float Compute** dialog box will close.
8. Now in the selected directory, there are two files, *.flt and *.hdr. The clipped digital terrain grid has been successfully converted to a binary format.

The grid can also be converted to ASCII file format for importing into HEC-FIA using **Raster to ASCII**. However, binary format is preferred due to smaller file sizes and faster processing time.

B.6   Preparing HAZUS Data

For a typical rapid consequence assessment, the required structure inventory will be generated using data that accompany FEMA's HAZUS-MH software. If better data exists in the form of a current structure inventory (point shapefile) or parcel database (polygon shapefile), those should be used. In order to use HAZUS county, census block and agricultural data, the geodatabase feature classes must be exported from the geodatabase to shapefile format.

1. Start **ArcCatalog**. The **ArcCatalog** window will open (see Figure B.1).
2. From the first pane, the **Catalog Tree**, browse to the folder that contains the HAZUS county and census block geodatabase file for the state that contains your study area (i.e., *bndrygbs.mdb*).
3. Double-click on *bndrygbs.mdb*, in the **Contents Pane** (Figure 3.33) to display the contents of the geodatabase file.
4. In the **Contents Pane**, hold down **CTRL** and click to select both **hzCensusBlock** and **hzCounty** (Figure B.27).
5. With both feature classes selected, right click on one of the features. From the shortcut menu that appears, point to **Export** and select **To Shapefile (multiple)** (Figure 3.34).

Contents	Preview	Metadata	
Name			Type
hzTract			Personal Geodatabase Feature Class
hzSqFootageOccupT			Personal Geodatabase Table
hzSqFootageOccupB			Personal Geodatabase Table
hzMeansCountyLocationFactor			Personal Geodatabase Table
hzExposureOccupT			Personal Geodatabase Table
hzExposureOccupB			Personal Geodatabase Table
hzExposureContentOccupT			Personal Geodatabase Table
hzExposureContentOccupB			Personal Geodatabase Table
hzDemographicsT			Personal Geodatabase Table
hzDemographicsB			Personal Geodatabase Table
hzCounty			Personal Geodatabase Feature Class
hzCensusBlock			Personal Geodatabase Feature Class
hzBldgCountOccupT			Personal Geodatabase Table
hzBldgCountOccupB			Personal Geodatabase Table
eqTractAttribs			Personal Geodatabase Table

Figure B.27 ArcCatalog Contents Pane.

6. The **Feature Class to Shapefile (multiple)** dialog box will open (Figure B.29). The two geodatabase feature classes that were selected in Step 6 are shown in the table under the **Input Features** box.

7. Browse to where you want to store the shapefiles by clicking 📂 next to the **Output Folder** box. The **Output Folder** browser will open (Figure 3.36). Browse to the folder where you want to store the shapefiles, click **Add**. The **Output Folder** browser will close, and the pathname with the selected folder will appear in the **Output Location** box of the **Feature Class to Feature Class** dialog box.

8. Click **OK**, the **Feature Class to Shapefile (multiple)** dialog box will close, and the **Feature Class to Shapefile (multiple) Compute** dialog box will open. This dialog box notifies the user of the status of the compute and how long it took to create the files. When finished viewing information, click **Close**, the **Feature Class to**
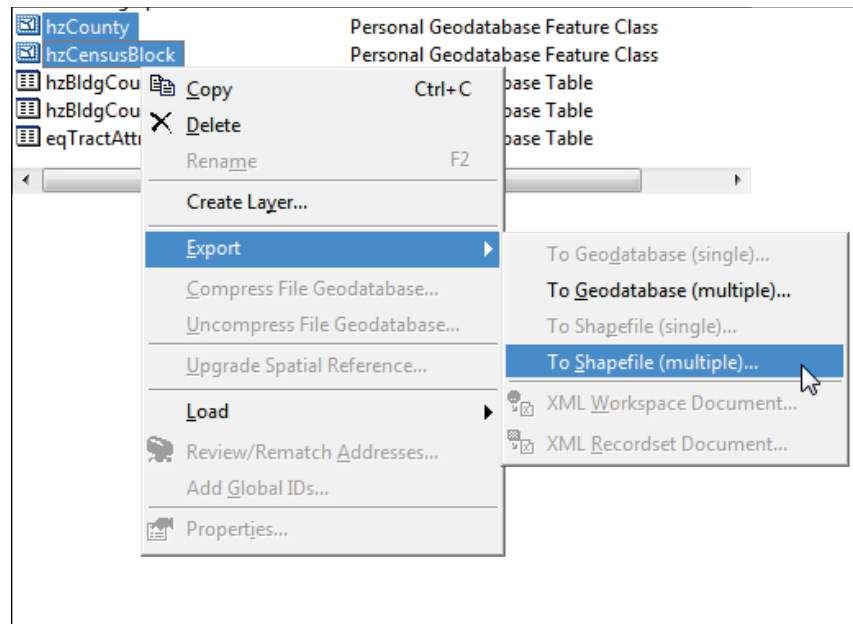
Figure B.28 Export Geodatabase to Shapefile (ArcCatalog Contents Pane).
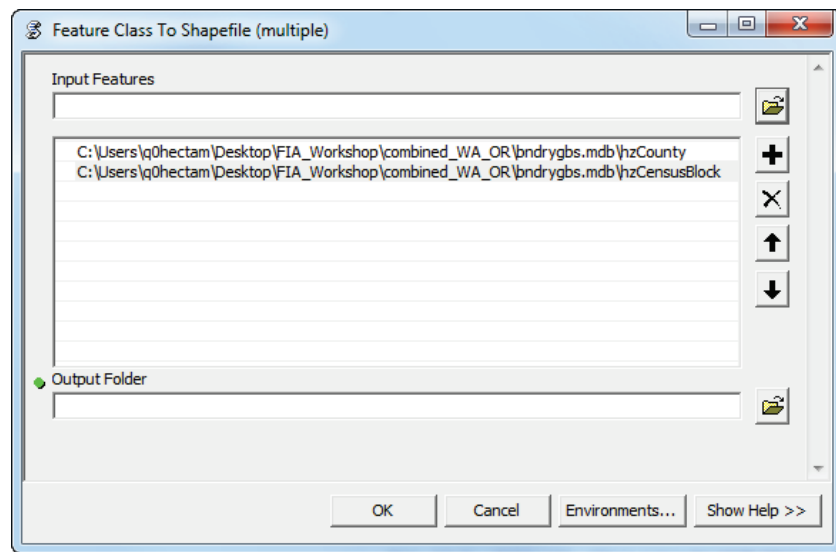
Figure B.29 Feature Class To Shapefile (Multiple) dialog box.

Shapefile (multiple) Compute dialog box will close and the multiple shapefiles (i.e., *hzCensusBlock.shp, hzCounty.shp*) will appear in the location you selected.

To select only the data that is needed for the study area, use the inundated area bounding polygon shapefile created in Section B.3 to clip the HAZUS shapefiles. Make sure the inundated area polygon used in this step represents the maximum possible water surface elevation (i.e., PMF with failure for a typical IES).

1. Start **ArcMap**. The **ArcMap** window will open (see Figure B.13).
2. You will need to add the two shapefiles generated from Step 9 (i.e., *hzCensusBlock.shp, hzCounty.shp*) and the inundated area bounding polygon shapefile created in Section B.3.
3.

From the **Table of Contents** pane of the **ArcMap** window, right-click

Layers, from the shortcut menu click **Add Data** ⬇ ; the **Add Data**

browser will open (Figure B.30). From the browser, locate the inundated area bounding polygon shapefile you wish to use and click **Add**. The **Add Data** browser will close and the selected shapefile will now appear in the **Table of Contents** pane under **Layers**.
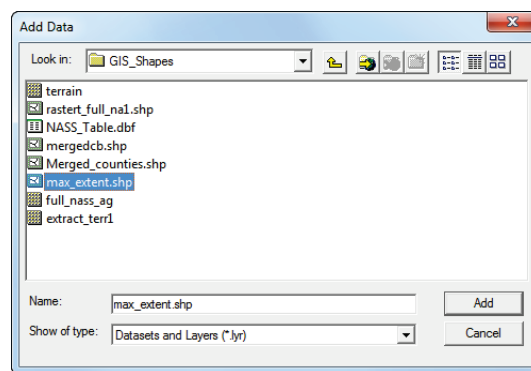
Figure B.30 Add Data browser.

4. From the **Selection** menu, click **Select by Location**. The **Select by Location** dialog box will open (Figure B.31).
5. From **I want to** drop-down list select "**select features from**". In the **following layer(s)** box click **hzCensusBlock**.
6. From **that** drop-down list select **intersect**. From **the features in this layer** drop-down list select the inundated area bounding polygon shapefile (i.e., *max_extent*).
7. Click **OK**, a **Selecting features** window will open. Once the software has selected all of the census blocks that are touched by the inundation area polygon, the **Select By Location** dialog box will close. On the display, the selected census blocks are displayed in cyan.
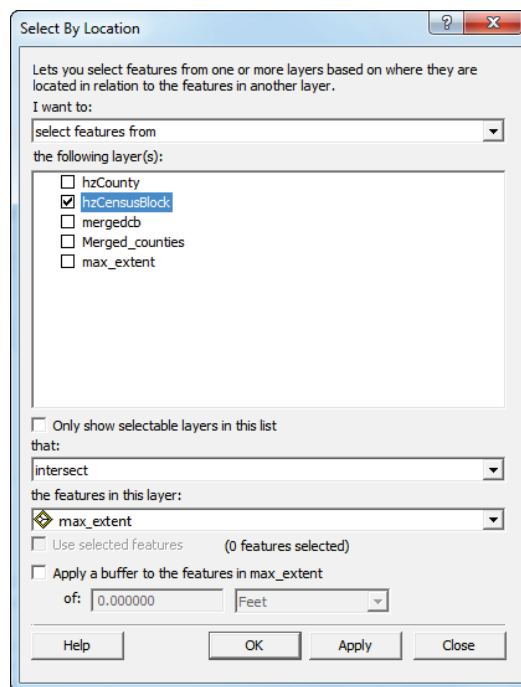
Figure B.31 Select By Location dialog box.

Next, export the selected polygons to a shapefile:

1. In the **Table of Contents** pane (Figure B.32), right click the census block shapefile (i.e., *hzCensusBlock*). From the shortcut menu point to **Data**, click **Export Data**; the **Export Data** dialog box will open (Figure B.33).
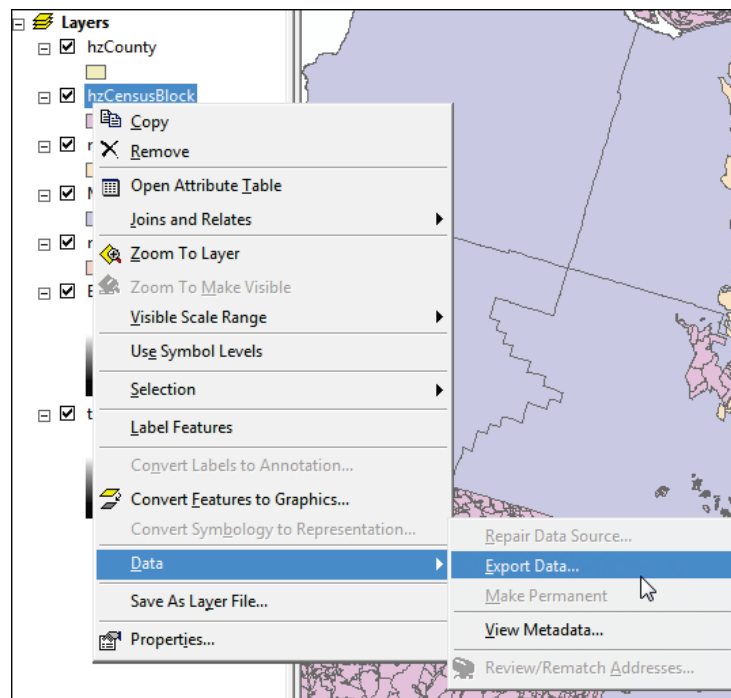
Figure B.32 Table of Contents pane.
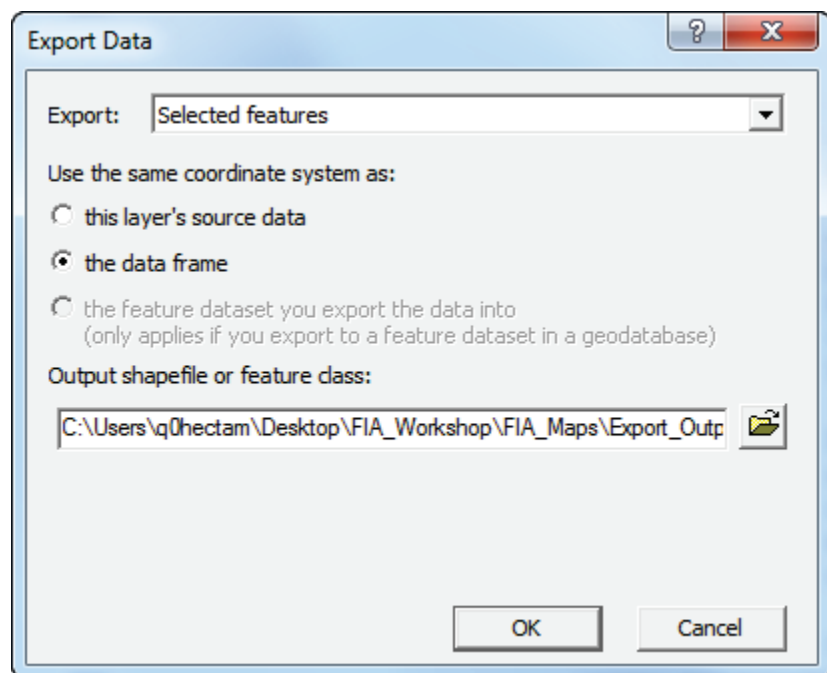
Figure B.33 Export Data dialog box.

2. From the **Export** drop-down list, select **Selected features**.
3. From **Use the same coordinate system as**, select **the data frame**.
4. Browse to where you want to store the shapefile by clicking [icon] next to the **Output shapefile or feature class** box. The **Saving Data** browser will open. Browse to the folder where you want to store the shapefile. The **Name** box will have a default name (i.e., *Export_Output.shp*); enter a name for the shapefile (i.e., *Truman_Censusclip.shp*). Click **Save**, the **Saving Data** browser will close, and the path and the shapefile name will appear in the **Output shapefile or feature class** box of the **Export** dialog box.
5. Click **OK**. An **Export Progress** window will appear, displaying the progress of exporting the data. Once the exporting of the data to the shapefile is finished, a window will appear asking you if want to add the shapefile to the map, click **No**.
6. Now you should have a shapefile of the clipped HAZUS data in the location you selected.

Repeat the above process of selecting only the data in the study area for the county data and export to a shapefile.

B.6.1   Using HAZUS Data for Multiple States

In addition to the census, county and agricultural GIS polygon datasets, the supporting HAZUS database(s) must be supplied to the person that will be running HEC-FIA. The folder containing the HAZUS data for a state within the study area in question should be copied to the HEC-FIA study folder for easy access during setup and running of HEC-FIA.

The HEC-FIA program uses one geodatabase for structure inventory and vehicle inventory. The two HAZUS geodatabases that are critical for use in HEC-FIA are named *bndrygbs.mdb* and *flVeh.mdb* and are separate for each state. If the study area crosses a state boundary, follow the same process outlined in the previous section for data in each state, then **append** the shapefiles for each data type (census blocks, counties, agricultural) into a single polygon before importing them into HEC-FIA. The supporting database tables must also be appended into one for each data type before importing into HEC-FIA. These database tables include the *hzBldgCountOccupB, hzExposureOccupB, hzExposureContentOccupB,* and *hzDemographicB* tables from the **bndrygbs.mdb** database, the *flNightVehicleInv* table in the **flVeh.mdb** database, and the *flAgricultureInventory* table from the **flAG.mdb** database.

Step-by-step instructions on how to append tables using ArcCatalog follows.

To merge HAZUS data for multiple states, do the following:

1. From the HAZUS data, locate your states of interest (i.e., **VA1** (Virginia), **WV1** (West Virginia)), these are folder names.
2. Copy the required folders to your study directory. It would be a good idea to create a backup copy of these folders somewhere else.
3. Start **ArcCatalog**. The **ArcCatalog** window will open (Figure B.1, page B-2).
4. If the **ArcToolbox Pane** is not present, click **ArcToolbox** on the **Standard Toolbar** (Figure B.2, page B-2) and the **ArcToolbox Pane** will display in the **ArcCatalog** window.
5. From the **ArcToolbox Tree**, click **Data Management Tools**, click **General**, and then double-click **Append**. The **Append** dialog box will open (Figure B.34).

6. Now select the HAZUS dataset that you want to be your input dataset. For example, the HAZUS geodatabase file for the state of Virginia will be used as the input dataset. From the **Input Datasets** box, click open . The **Input Datasets** browser will open. Select the HAZUS geodatabase file for the structure inventory (i.e., *bndrygbs.mdb*) for the state you are interested in (i.e.,

Figure B.34 Append dialog box.

VA1) click **Add**. From the list, select the appropriate personal geodatabase table (i.e., *hzBldgCountOccupB*), click **Add**. The **Input Datasets** browser will close and the name will appear in the **Input Datasets** table on the **Append** dialog box.

7. Now select the HAZUS dataset that you want to be your target dataset (which is your master file). For example, the HAZUS geodatabase file for the State of West Virginia will be used as the target dataset. From the **Target Datasets** box, press open. The **Target Dataset** browser will open. Select the HAZUS geodatabase file for the structure inventory (i.e., *bndrygbs.mdb*) for the state you are interested in (i.e., *WV1*) click **Add**. From the list, select the appropriate personal geodatabase table (i.e., *hzBldgCountOccupB*),

click **Add**. The **Target Dataset** browser will close and the name will appear in the **Target Dataset** box on the **Append** dialog box.

8. Now the **Append** dialog box should have information about the input dataset in the **Input Datasets** table and information about the target dataset in the **Target Dataset** box. Click **OK**, the **Append** dialog box will close.

9. The Append compute dialog box will open. The compute process time is dependent on the amount of information contained in the geodatabase files. Once the computation has finished, click **Close**. The **Append** compute dialog box will close.

10. For the structure inventory, there are three other personal geodatabase tables that need to be merged - *hzDemographicsB*, *hzCensusBlock*, and *hzExposureOccupB*. Repeat Steps 5 through 9 to complete the merging.

11. For the vehicle inventory (i.e., *flVeh.mdb*) repeat Steps 5 through 9, selecting the *flNightVehicleInv* personal geodatabase table.

After appending all the appropriate files and tables it will be possible to import the target dataset bndrygbs.mdb and flVeh.mdb files into the FIA structure inventory. The census block shapefiles must also be merged into one shapefile prior to importing them into HEC-FIA. For this example, if census blocks exist in one shapefile for the inundated areas within the states of Virginia and West Virginia, then the bndrygbs.mdb file for West Virginia (the target dataset above) would be selected during the Generate Inventory using HAZUS database option.

B.7   Preparing HEC-RAS Data

HEC-FIA requires a stream centerline for all consequence computations. The life loss computation in HEC-FIA also requires a cross-section shapefile, storage area shapefile, and associated HEC-DSS database with stage-hydrograph output at each cross-section and storage area for the flood scenarios being assessed.

B.7.1   Stream Centerline, Cross-Section, Storage Area

The stream centerline, cross-section, and storage area feature classes are stored in the HEC-GeoRAS geodatabase. To create a shapefile of the stream centerline:

1. Start **ArcCatalog**. The **ArcCatalog** window will open (see Figure B.1, page B-2).

2. From the first pane, the **Catalog Tree**, browse to the folder that contains the HEC-GeoRAS geodatabase for your study.

3. Once the geodatabase file appears in the **Contents** tab of the main window (), double-click on the geodatabase filename (i.e.,

Turman_SDF.mdb). Now double-click on the geodatabase feature dataset (i.e., *RasResults*).

4. The stream centerline file created by HEC-GeoRAS should appear in the list (i.e., *River2D*). Right-click on the stream centerline filename (see Figure 3.9, page 3-4), from the shortcut menu point to **Export**, click **To Shapefile (single)**.

5. The **Feature Class to Feature Class** dialog box will open (see Figure B.10).

6. The **Input Features** box (see Figure B.10) should contain the name of the selected stream centerline file. Browse to where you want to store the shapefile by clicking 📂 next to the **Output Location** box (see Figure B.10). The **Output Location** browser will open (see Figure B.11). Browse to the folder where you want to store the shapefile, click **Add**. The **Output Location** browser will close, and the pathname with the selected folder will appear in the **Output Location** box of the **Feature Class to Feature Class** dialog box.

7. In the **Output Feature Class** box (Figure B.10), provide a name for the shapefile.

8. Click **OK**. The **Feature Class to Feature Class** dialog box will close and the **Feature Class to Feature Class Compute** dialog box will open (see Figure B.12). This dialog box will contain information on start time, if creation was successful, end time, and how long it took to create the binary file. Click **Close**, the **Feature Class to Feature Class Compute** dialog box will close and the shapefile will appear in the location you selected.

9. Now in the selected directory, the shapefile will appear. The stream centerline has been successfully exported to a shapefile.

10. Repeat Steps 1-10 for the storage area (i.e., *StorageAreas*) shapefile (i.e., *Truman_StorageAreas*) and the cross sections (i.e., XSCutlines) shapefile (i.e., *Truman_XSections*).

If HEC-GeoRAS and HEC-RAS were not used to develop the hydraulic results for the study, the stream alignment in HEC-FIA can be imported from an existing line shapefile or digitized using tools in HEC-FIA. Clip the stream centerline to the extents of the study area using the steps described below. Economic consequence can be computed in HEC-FIA using any stream alignment and depth grid as input. However, HEC-FIA can not compute loss of life consequences unless a cross-section shapefile, storage area shapefile, and associated HEC-DSS database are provided as input, which are output from HEC-RAS and HEC-GeoRAS.

B.7.2   HEC-DSS File

For life loss computation, HEC-FIA can obtain flood wave arrival times for individual structures from arrival time grids *or* from stage-hydrographs stored by

HEC-RAS in an HEC-DSS (Hydrologic Engineering Center's Data Storage System) file. The hydraulic engineer will need to configure HEC-RAS to generate stage-hydrographs at all cross-section and storage area locations before simulating an event. If the HEC-DSS file already contains hydrograph data for each cross-section, skip this section.

1. Start **HEC-RAS**. The **HEC-RAS** window will open (Figure B.35).
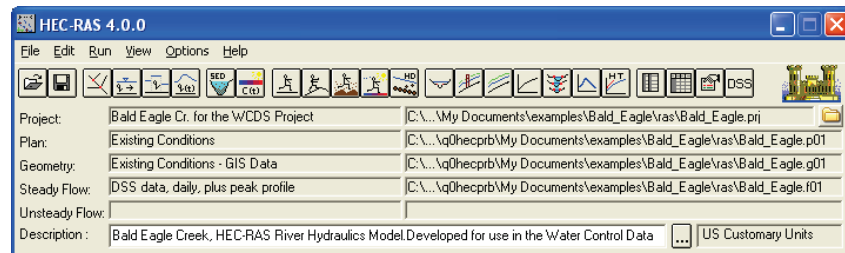
Figure B.35 HEC-RAS Main Window.

2. From the toolbar click , the **Unsteady Flow Analysis** dialog box will open (Figure B.36).
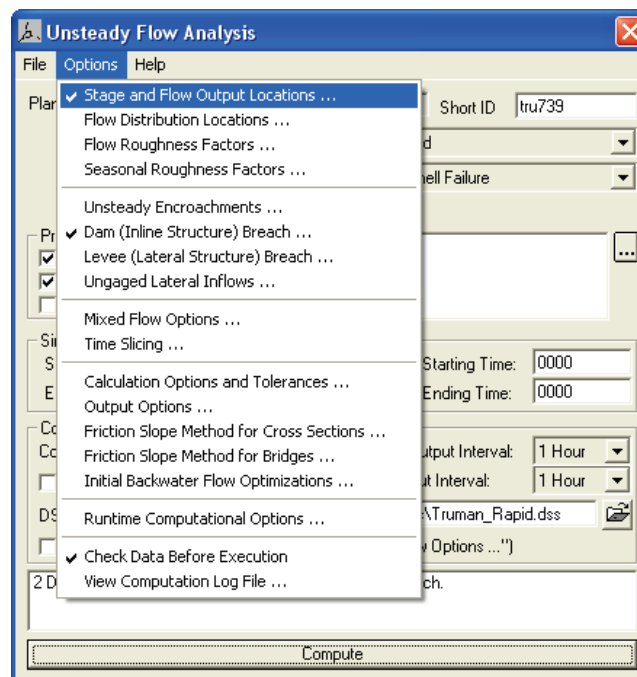
Figure B.36 Unsteady Flow Analysis dialog box.

3. From the **Options** menu, click **Stage and Flow Output Locations**, the **Stage and Flow Output Locations** dialog box will open (Figure B.37).
4. From the **River** drop-down list, select **(All Rivers)**.
5. From the **RS** box, select **(All Rivers)**.
6. Click the arrow to select all the cross-section locations to the **Selected Locations** box.
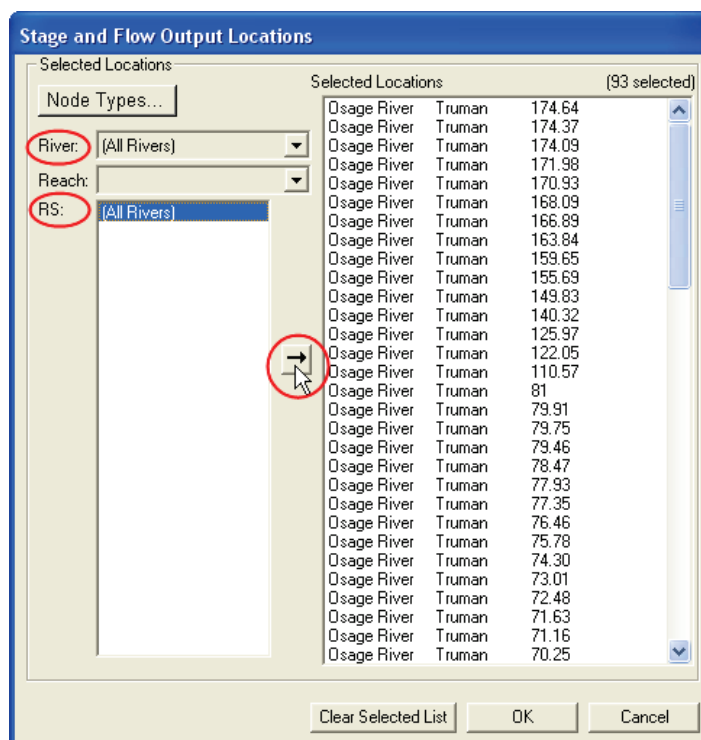
Figure B.37 Stage and Flow Output Locations dialog box.

7. Click **OK**, the **Stage and Flow Output Locations** dialog box will close.

8. From the **Unsteady Flow Analysis** dialog box (see Figure B.36), in the **DSS Output Filename** box, enter the path and the filename of the DSS file that will contain the stage-hydrographs created by HEC-RAS.

9. Click **Compute**, the **Unsteady Flow Analysis** dialog box will close, and the **HEC-RAS Computations** dialog box will open. When computations are finished, click **Close**, the **HEC-RAS Computations** dialog box will close.

10. The HEC-DSS file will be created in the location that you selected.

B.8   Impact Areas

Impact areas are required by HEC-FIA. They are used for defining common input parameters for groups of structures as well as for aggregating consequence results. A county dataset (shapefile) is often the most readily available dataset for defining impact areas. When county boundaries are used as impact areas, damages can be reported by individual structure, county, or larger area.

Impact areas are polygons and the polygon shapefile can contain several discrete polygons, but HEC-FIA cannot import multi-part polygons. So each individual polygon must have a different name.

Geo-referenced county boundaries are available in the HAZUS dataset for all states. Section B.6 outlines the steps necessary for preparing HAZUS data for use in HEC-FIA, so for the impact areas use the county shapefile (i.e., *TrumanCounties*) that was created.

If a geo-referenced polygon shapefile is not available for use as impact areas, or if some polygons do not import because they are multi-part polygons, they can be digitized in HEC-FIA. For further information on digitizing impact areas in HEC-FIA, refer to Chapter 5, Section 5.2.5 (page 5-23).

B.9   Double Warning Data

For flood scenarios where warnings would go out to different parts of the inundated area at different times, the double-warning approach may be necessary. For example, for any dam failure flood scenario where the non-fail releases from the dam would cause significant damage (i.e. most PMF events), the double-warning approach is required. The double-warning approach in this example would allow HEC-FIA to warn those people impacted by the non-fail inundation at a different time (earlier) than those impacted by the dam failure inundation.

Follow these steps to prepare the data necessary for the double-warning procedure. The example uses full pool failure (FP_Fail) and full pool non-fail (FP_NonFail) inundations.

1.   Open **ArcMap** (Figure B.13), and add the FP_Fail and FP_NonFail shapefiles using the ⬇ **add data** button. If the **ArcToolbox Pane** is not present, click **ArcToolbox** 🧰 on the **Standard Toolbar** (see Figure B.2)

2.   From the **ArcToolbox Tree**, double-click **Analysis Tools**, double-click **Overlay**, and then double-click **Union** (Figure B.38). The Union dialog box will open (Figure B.39).

3.   Use the drop down list next to the **Input Features** box (Figure B.39). Select the FP_Fail and FP_NonFail shapefiles. They will appear in the table below the input features box after they are selected.

4.   Change the default name in the **Output Feature Class** (Figure B.39) to reflect that it is a union of the two files.

Figure B.38 ArcToolbox tree.

5.   Leave all other options as their default and press **OK**. The **Union** dialog box will close and the **Union Dialog Compute** box will open indicating when the union tool has completed its task
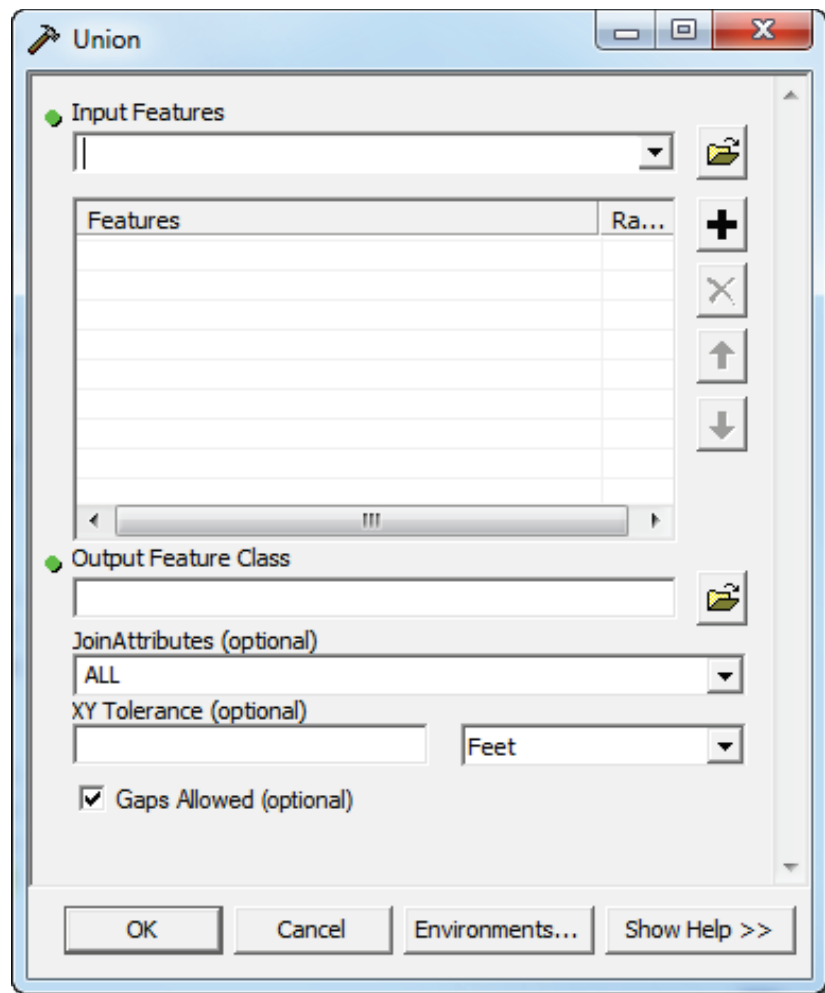
.
Figure B.39 Union dialog box.

6. Open the attribute table of the new shapefile by right clicking on it in the table of contents pane. Add a new field to the table by pressing **Options** and then clicking on **Add Field**. The **Add** Field Dialog box will appear. In the **Name** box enter "Name", in the **Type** box choose "Text" from the dropdown list, and in the **Field Properties** section enter 20 for **Length** as shown in Figure B.40. Press **OK**.
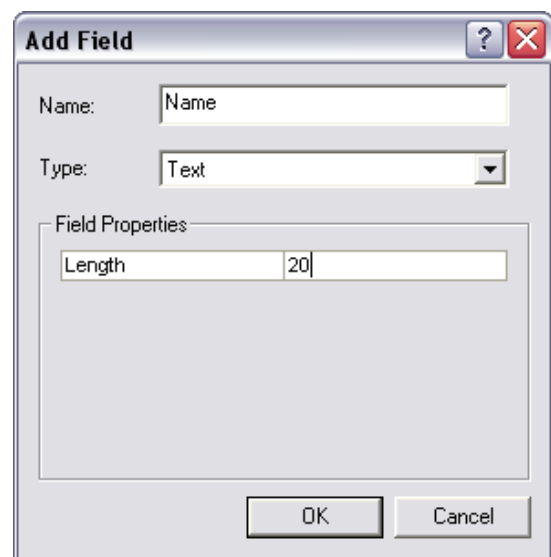
Figure B.40 Add Field dialog box

7. Next, enter the appropriate name for each polygon feature (FP_NonFail, FP_Fail). From the **Editor** toolbar, select **Start Editing** (see Figure B.19, page B-13).

8. Double-click in the appropriate cell of the attribute table and enter the correct name for each polygon in the table (Figure B.41).
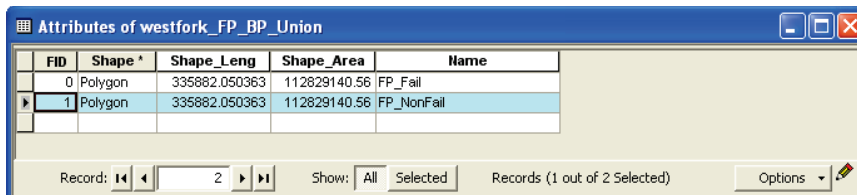
Figure B.41 Entering names into the attribute table.

9. From the editor toolbar, select **Stop Editing** (Figure 3.45). Save the edits. The double warning time polygon shapefile (*westfork_FP_BP_Union.shp*) is now ready for use in HEC-FIA.

B.10   Agricultural Grid Preparation

NASS CDL grids are generally the preferred data source for quick assessments of agricultural damage. The NASS CDL agricultural data is available online for all continental US States at the following website: http://www.nass.usda.gov/research/Cropland/SARS1a.htm.

The NASS agricultural grids are in .tif file format. The projection information is contained within a world file (.tfw). Follow these steps to prepare the agricultural data for use in HEC-FIA:

1. Open **ArcMap** (see Figure B.13, page B-9), add the maximum inundation shapefile (i.e., *max_extent.shp*) obtained in Section B.3 using the ✦ **add data** button.

2. Next, add the NASS agricultural grid file.

3. Not all data in the NASS CDL layer are required; only those within the study area should be selected. Right-Click on the shapefile in the table of contents and click **zoom to layer** (Figure B.42). This will zoom the map window to the shapefile extent.
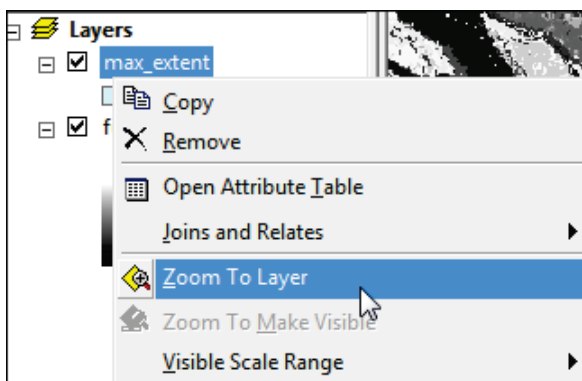
Figure B.42 Zoom to Layer Menu Option.

4. Next, Right-Click on the NASS CDL grid in the table of contents and select **Data → Export Data** (Figure B.43).
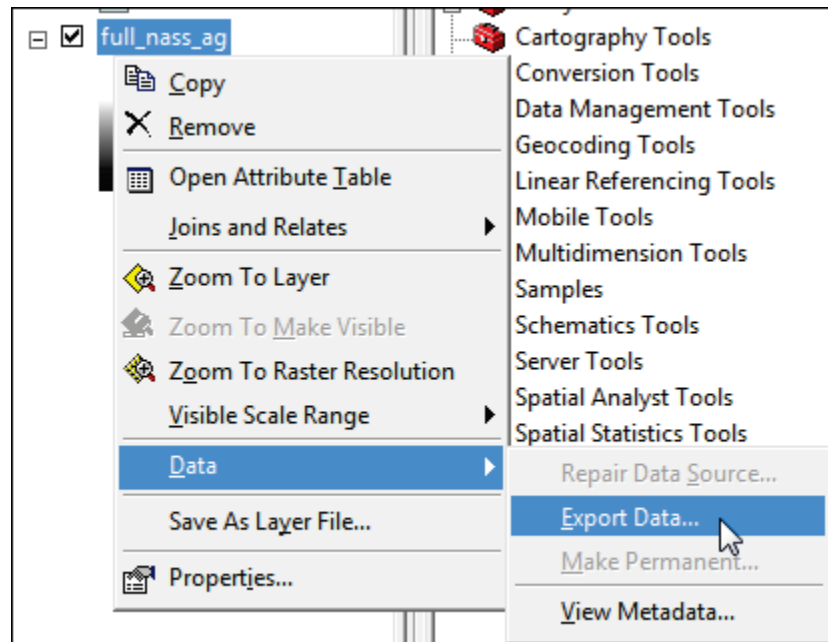
Figure B.43 Export Data Layer Menu Option

5. The Export Raster Data Dialog box will appear (Figure B.44).
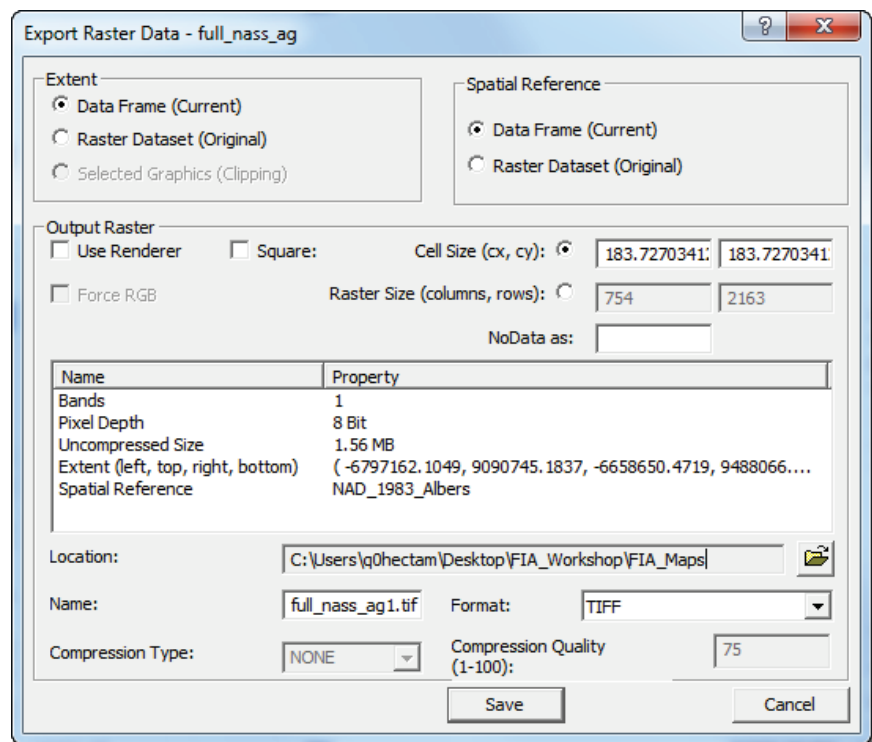
Figure B.44 Export Raster Data Dialog

6. Choose **Data Frame (Current)** for the Extent. This will export the data to the current frames extent, which is based on the maximum inundation shapefile set in step 3.

7. Since the shapefile was added first, the data frame should be using the required spatial reference. Choose **Data Frame (Current)** for the Spatial Reference. This will project the NASS CDL data into the same projection as the maximum inundation shapefile, which should be the projection system used for all FIA GIS data.

8. Leave the middle portion of the dialog with the default values. The Cell Size(cx, cy) is automatically adjusted to meet the selected spatial reference.

9. Browse to a location to store the shapefile by clicking 🖿 next to the **Location** box (Figure B.44).

10. Next choose **TIFF** from the dropdown list of the **Format** box. In the **Name** box, enter the name of the grid that is to be exported.

11. Press the **Save** button. The Export Data Dialog box will close and you will be prompted to add the new data to your ArcMap project. Select No, the prompt will close, and the exported data will not be added to your ArcMap project.

Figure B.45 Export Raster World File.

12. Finally, create a world file for the exported data.

13. From the **ArcToolbox Tree** (see Figure B.3), double-click **Data Management Tools**, double-click **Raster**, then **Raster Properties**, and then double-click **Export Raster World File** (Figure B.45)**.** The **Export Raster World File** dialog box will open (Figure B.46).
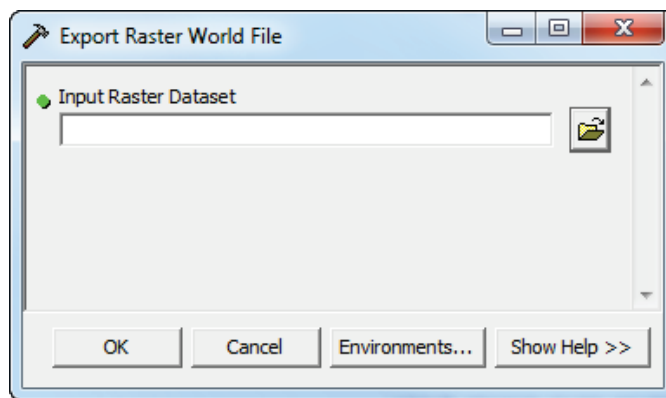
Figure B.46 Export Raster World File dialog box.

14. Browse to the location of the masked digital terrain grid file by clicking open 📂 next to the **Input raster** box.

15. Once the exported .tif file is selected (i.e., *westfork_NASS_ag.tif*), click **OK.** The associated world file (.tfw) will be created in the same directory as the .tif file. The Agricultural grid is now ready to be imported in HEC-FIA.

Appendix C

Individual Project Benefit Allocation

Each year, the U.S. Army Corps of Engineers (USACE) provides a report to Congress, which includes flood damage prevented, flood damage incurred, potential damage, and some accumulative values of total benefits versus total investments. These values are reported on a state by state basis. HEC-FIA can help calculate these values. While not reported to Congress, the USACE Headquarters office requests that the Districts supply individual project benefits.

C.1   Overview

FIA can allocate the overall benefits to individual projects (levees and reservoirs). The benefits computed for each impact area can be distributed to the projects (local and remote) that are protecting the impact area. The allocation of project benefits can be performed automatically by routing the reservoir holdouts, and/or they can be entered manually.

When developing the FIA procedure, the Corps' current methods for performing individual project benefit accomplishment calculations were considered. Because many of the districts were developing their allocations differently, the need for automating this process for consistency, efficiency and practicality was recognized. The procedure also considers the level of accuracy inherent with the program's other data entries.

A chronological list of the projects is not used with this allocation process. Projects are incrementally justified when they are approved, and therefore a project is not handicapped because it may have been added at a later date. The project is part of the system and is considered as such.

Ordinarily, the allocation of individual project benefits is only going to be performed as a post-event process. During the event, or in the forecasted mode, the calculation of individual project accomplishments is not necessary. For immediate reporting purposes, such as during real time or forecasted events, total damages and total benefits are still calculated by FIA.

C.2 Procedure

In FIA, reservoirs receive the initial credit and levees receive any remaining benefits. This sequencing of benefits appeared to be widely accepted by the Corps' district offices. In most cases, levees would still receive sizable benefits because flows contributed by uncontrolled drainage areas and reservoir releases would still put water on the levee. Figure C.1 illustrates the distribution of the benefits to the reservoirs and levees.
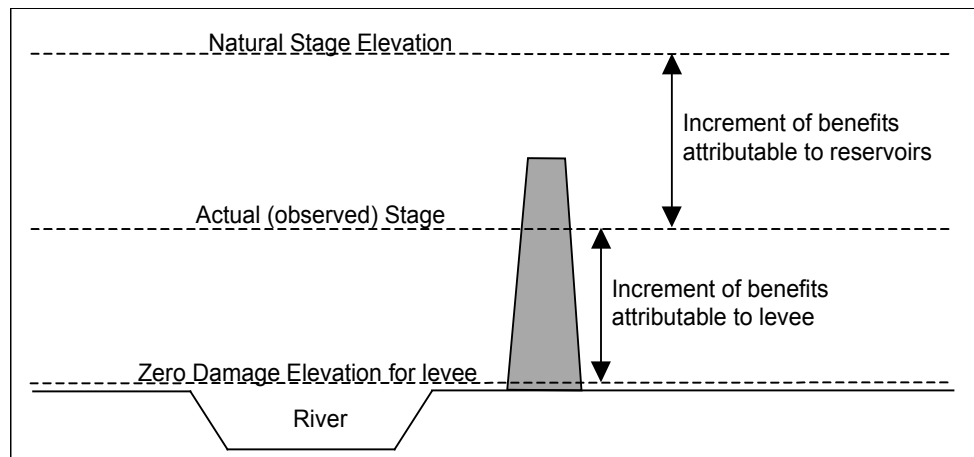
Figure C.1 Distribution of Project Benefits.

C.2.1 Computation of Total Benefits

Benefits are calculated as the difference between the damage that would occur without reservoir regulation and without levee protection and damage that would occur with reservoir regulation and with levee protection. In order for FIA to calculate total benefits, the user must be able to provide regulated (observed) and unregulated (natural) flow or stage hydrographs for the events they want to analyze. These hydrographs are required at all hydrograph locations. The hydrographs may come from programs such as HEC-ResSim or gaged records. When using flow hydrographs, rating curves must also be supplied at each hydrograph location. For more accurate benefit accounting, it is preferable to provide both with-levee and without-levee rating curves at each hydrograph location. If only one rating curve is entered, FIA will use that curve for all damage computations.

To compute reservoir and levee benefits, FIA performs three damage computations:

- Without-project damage (i.e. no levees or reservoirs),
- Without-levee damage, and
- With-project damages.

Without-project damage is the damage that would be caused by an event if the reservoirs and levees did not exist. FIA computes this damage using unregulated (natural) flows and without-project rating curves (if available) supplied by the user to determine the stage next to an impact area. The damage in that impact area is found on the stage-versus-damage relationship for that impact area. It is best to use the without-project rating curve in this case because, for a given flow that overflows the banks of a river, the stage in the river may be higher if levees are in place than if no levees exist. If the user supplies stage hydrographs instead of flow hydrographs, rating curves are not needed.

Without-levee damage is the damage that would be caused by an event if the levees did not exist. Since the reservoir exists, FIA computes this damage using regulated (observed) flows and, if available, without-project rating curves supplied by the user.

With-project damage is the damage that is caused by an event given the existing system, including reservoirs and levees. FIA computes this damage using regulated (observed) flows and the with-levee rating curve.

FIA computes total reservoir benefits by subtracting the without-levee damage from the without-project damage. It then computes total levee benefits by subtracting with-project damage from without-levee damage. Reservoir benefits are then assigned to individual projects based on reservoir holdouts (automatic method) or user specified percentages (manual method). Each method is discussed in Section C.2.2.

Levee benefits are assigned directly to the levee protecting a given impact area. The percentage of levee benefits is computed as the dollars saved by the levee versus the total damage prevented for the impact area. This process assumes that the levees are not overtopped or do not fail in the with-project condition. If a levee is overtopped or fails, then is receives no benefit for this event.

C.2.2   Assigning Benefits to Individual Reservoirs

Automatic Method

If the user elects to have the program automatically allocate the benefits, the regulated (with-project) flow or stage hydrograph, assigned during the definition of the hydrograph time-series set, is used along with the results of HEC-ResSim (Reservoir Evaluation System Simulation) to allocate the benefits. HEC-ResSim is used because it readily computes regulated and unregulated flows and routes reservoir holdouts. Additionally, like FIA, it is a component of the Corps Water Management System (CWMS) suite of software.

For a given impact area, the automated procedure allocates the total benefit among the reservoirs based on the peak flow reduction associated with each reservoir. The reservoir's peak flow reduction is computed through the routing of each reservoir's flow holdouts. These flow holdouts are routed to the hydrograph locations. The development and routing of holdouts is performed in the following manner: ResSim writes reservoir storage-versus-elevation time-series for each reservoir, for each event, to DSS. ResSim uses the reservoir's storage data, converts incremental storage into flow, and routes these holdout flows to the hydrograph locations. ResSim uses the same routing techniques as it did for the regulated and unregulated flows. The routing method is defined in ResSim. Next, the routed holdouts are saved to DSS. When requested to perform project benefit accomplishment, FIA uses the DSS records to add all the routed flows at each of the hydrograph locations at the time of the unregulated peak for that node. The sum of the routed flows from the appropriate reservoirs should be the total difference in regulated and unregulated peak flows. If necessary, an adjustment to this sum is made to make sure the sum of the holdouts equals the total difference in the regulated and unregulated peak flows. Any error is equally proportioned among all the holdouts. Next, the program divides the individual holdout flows from each of the reservoirs by the total holdout it just calculated to find the percentage of flow reduction associated with each reservoir above a given hydrograph location. This percentage of flow reduction is the percentage of total benefits for each reservoir. This percentage is used for all impact areas associated with that hydrograph location. The real advantage to this method is that it is completely automated. No intermediate runs have to be performed and a good portion of the data is already or readily available.

During the development of this procedure, a basic question had to be answered. Should the regulated or unregulated peak flow time be used for the calculation of benefits? The time of the peak must be known so FIA knows when to add the routed holdouts. FIA uses both peaks to calculate the total change in stage and that change in stage is what is used for the calculation of total benefits. However, what peak (regulated or un-regulated) would be more appropriate to use so that the combined routed holdouts from each reservoir would add up to the difference in total peak discharge? Regardless of which peak is chosen, there is likely to be some error in comparing the addition of the holdouts to the total difference in peak discharge. This procedure uses the time of the unregulated peak. Later, FIA makes adjustments to proportion the routed holdouts so they add to the total sum of the difference.

Manual Method

If the user chooses to assign reservoir benefits manually in the **Impact Area Editor** (see Project Benefits, page 9-13), they must enter reservoir allocation percentages. A table is provided in the **Edit Time Series**

Mappings dialog box to allow for reservoir allocation percentage entry (see Chapter 9, Section 9.8.2, page 9-43). Presumably these percentages are known or are mandated by some agreement. Individual reservoir benefits are then allocated to the reservoirs via the assigned percentages.

C.3   Levees

To automate the percentage of levee benefits, FIA takes the regulated discharge (observed flow), converts the flow to a stage using the rating curve supplied in FIA, and then compares the stage to the damage versus stage relationship (damage function) in FIA. The remaining damage reduction benefits are attributed to the levee. The percentage of levee benefits is computed as the dollars saved by the levee versus the total dollars saved by the entire system. This process assumes that the levees are not overtopped or do not fail in the with-project condition. If a levee is overtopped or fails, then it receives no benefit for this event. Please see Figure C.1 for a physical depiction of how much benefit a levee receives.

Appendix D

Coordinate System Setup

To maintain a geographic reference (also called a geo-reference), FIA uses a user-selected **coordinate system**, called the World Coordinate System (WCS). This superimposes a grid on layer features to establish x- and y-coordinates in WCS for each point on the layer. The x-coordinate is referred to as "easting" and the y-coordinate is referred to as "northing" following customary use in surveying and mapping. The user can select the extent of this grid, the dimensions of the cells of the grid, the units of measurement of the grid, and even the location of the origin of the grid. This appendix focuses on how to setup and edit different coordinate systems.

D.1   Geographic Referencing

To maintain a geographic reference, you must specify a coordinate system for each FIA study. To establish the grid size and coordinate system:

1. From the **Menu Bar**, point to the **Maps** menu and select **Default Map Properties** (Figure D.1).
2. The **Map Default Properties** dialog box will open (Figure D.2).
3. The information in the **Map Default Properties** dialog box includes:
   Coordinate System: this box identifies the established coordinate system for a watershed. To edit the coordinate system, click **Edit**.
   Extents – Easting Min, Max, Northing Min, and Max. These values indicate the location of the left, top, right, and bottom borders of the grid in the display area. For example, the coordinates corresponding to the geographic extents specified in Figure D.2 are labeled in Figure D.3.
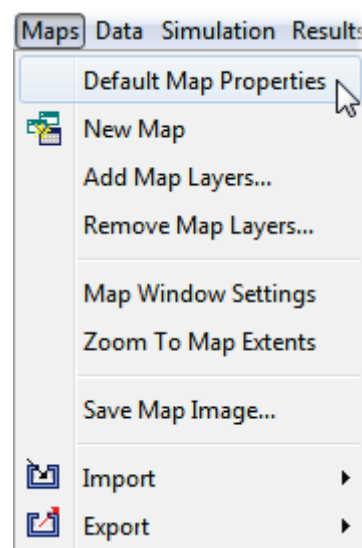
Figure D.1 Maps Menu (Menu Bar).
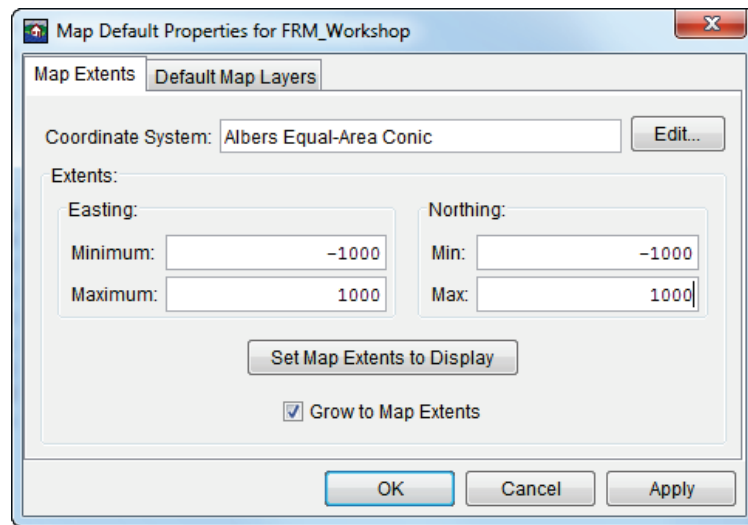
Figure D.2 Map Default Properties dialog box.
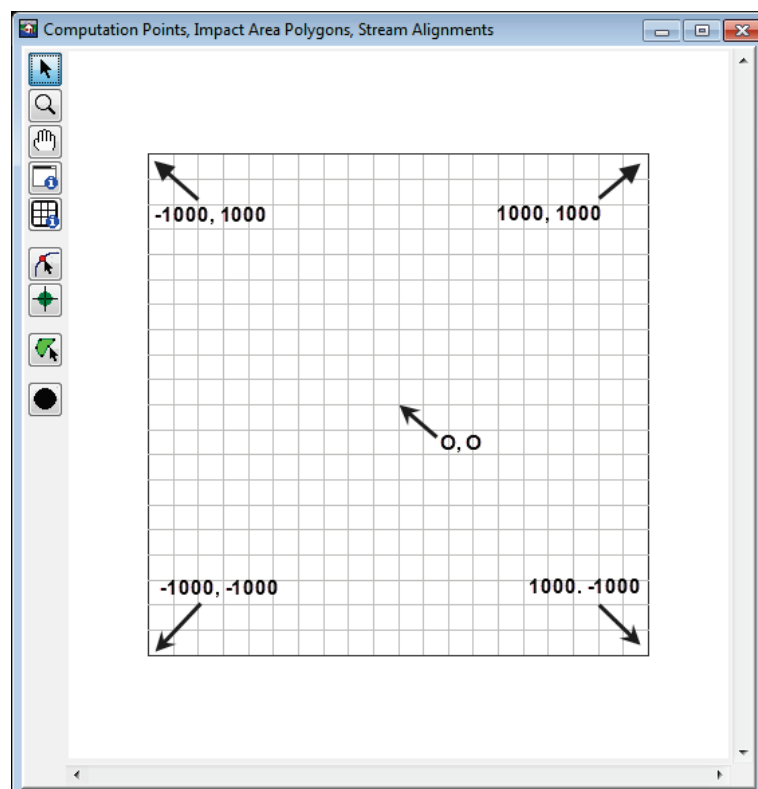
Figure D.3 Example of Geographic Extents and Equivalent Coordinates.

Grow to Map Extents. When this check box is selected, FIA automatically sets the geographic extents to define the smallest rectangle that encompasses all the objects in the workspace.

Note that the geographic extents of your layers must be selected carefully to ensure that the entire study area is included. Further, the extents specified and the coordinate system used must be consistent for all the

layers. Therefore, GIS tools are necessary to transform the layers from one coordinate system to another **before** using them with FIA. FIA does not have any coordinate system transformation tools.

D.2   Coordinate System Types

A **coordinate system** is a method of representing points in a space of given dimensions by coordinates. Two coordinate system types of interest to FIA are the geographic coordinate system, which is based on latitude and longitude coordinates, and the project coordinate system, which represents the projection of a geographic coordinate system on a plane.

The coordinate systems available in FIA are shown in Table D.1. This table also shows the units and spheroid type for each coordinate system. A **spheroid** (ellipsoid) is the shape of the earth that used in the calculations that transform positions on the curved surface of the earth to positions on a flat map. It is part of the horizontal datum, which approximates the curved surface of the earth over part of the globe.

Table D.1 Coordinate systems available in FIA.

Coordinate Systems	Units	Spheroid
X-Y	U.S. feet	N/A
Geographic	radians	WGS 72
Universal Transverse Mercator	meters	GRS 1980(NAD83)
State Plane Coordinates	meters	GRS 1980(NAD83)
Albers Equal-Area Conic	Degrees of Arc	Sphere of Radius 6370997 meters
Lambert Conformal Conic	International Feet	
Transverse Mercator	U.S. feet	Clarke 1866(NAD27)
Albers Equal-Area Conic (SHG)	meters	GRS 1980(NAD83)
Polar Stereographic (HRAP)	meters	Sphere of Radius 6371200 meters

D.2.1   X-Y Coordinate System

The X-Y coordinate system is also known as the World Coordinate System (WCS). This superimposes a grid on layer features to establish x- and y-coordinates for each point on the layer. The origin point (0,0) can be located anywhere, but every object is located in relation to the origin point. There is no spheroid specification for an X-Y coordinate system.

To set parameters for the X-Y Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.4).

3.   From the **System** list, select *X-Y*.
4.   From the **Units** list, select the units for the X-Y coordinate system.
5.   Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, X-Y should be displayed.
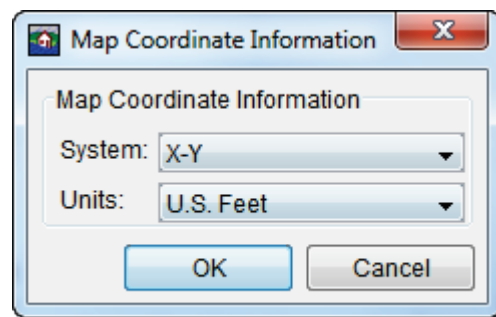
Figure D.4 X-Y Coordinate System— Map Coordinate Information dialog box.

D.2.2   Geographic Coordinate System

Each of the different coordinate systems will have their own set of options editable in the **Map Coordinate Information** dialog box. For a Geographic Coordinate System, users can define **units** and **spheroid**.

To set parameters for the Geographic Coordinate System:

1.   On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2.   The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.5).
3.   From the **System** list, select *Geographic*.
4.   From the **Units** list, select the units for the *Geographic* coordinate system. The only units available should be **Radians, Degrees of Arc**, and **Seconds of Arc**.
5.   From the **Spheroid** list, select a spheroid type for the *Geographic* coordinate system.
6.   Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Geographic* should be displayed.
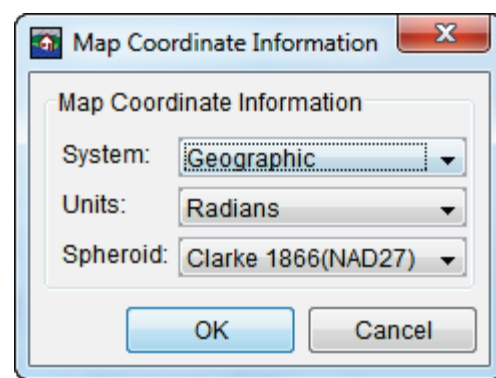
Figure D.5 Geographic Coordinate System.

D.2.3 Universal Transverse Mercator Coordinate System

The Universal Transverse Mercator (UTM) coordinate system is a projection coordinate system. UTM is used to define horizontal positions throughout the world by dividing the surface of the Earth into six-degree zones, with a central meridian in the center of each zone.

To set parameters for the Universal Transverse Mercator Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.6).
3. From the **System** list, select **Universal Transverse Mercator**.
4. From the **Units** list, select the units for the *Universal Transverse Mercator* coordinate system.

5. From the **Spheroid** list, select a spheroid type for the *Universal Transverse Mercator* coordinate system.

Figure D.6 Universal Transverse Mercator Coordinate System.

6. In the **UTM Zone** box, enter the UTM zone number.
7. Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Universal Transverse Mercator* should be displayed.

D.2.4 State Plane Coordinate System

The State Plane Coordinate System (SPCS) was established in the 1930's and now covers all fifty states. Zones were established for each state using either the Lambert Conformal or Traverse Mercator projections. Units are generally in feet (NAD27) or meters (NAD83).

To set parameters for the State Plane Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.7).
3. From the **System** list, select **State Plane Coordinates**.
4. From the **Units** list, select the units for the State Plane coordinate system.
5. From the **Spheroid** list, select a spheroid type for the State Plane coordinate system. There should only be two choices: **Clarke 1866(NAD27)** and **GRS 1980(NAD83)**.

Figure D.7 State Plane Coordinate System.

6. In the **Zone** box, enter the zone for the state. These codes are FIPS codes.
7. Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *State Plane* Coordinates should be displayed.

D.2.5   Albers Equal-Area Conic Coordinate System

The Albers Equal-Area Conic projection was established for mapping large areas. The projection is an equivalent projection, where areas are proportional and directions are true in limited areas.

To set parameters for the Albers Equal-Area Conic Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.8).
3. From the **System** list, select **Albers Equal-Area Conic**.
4. From the **Units** list, select the units for the Albers Equal-Area Conic coordinate system.
5. From the **Spheroid** list, select a spheroid type for the Albers Equal-Area Conic coordinate system.
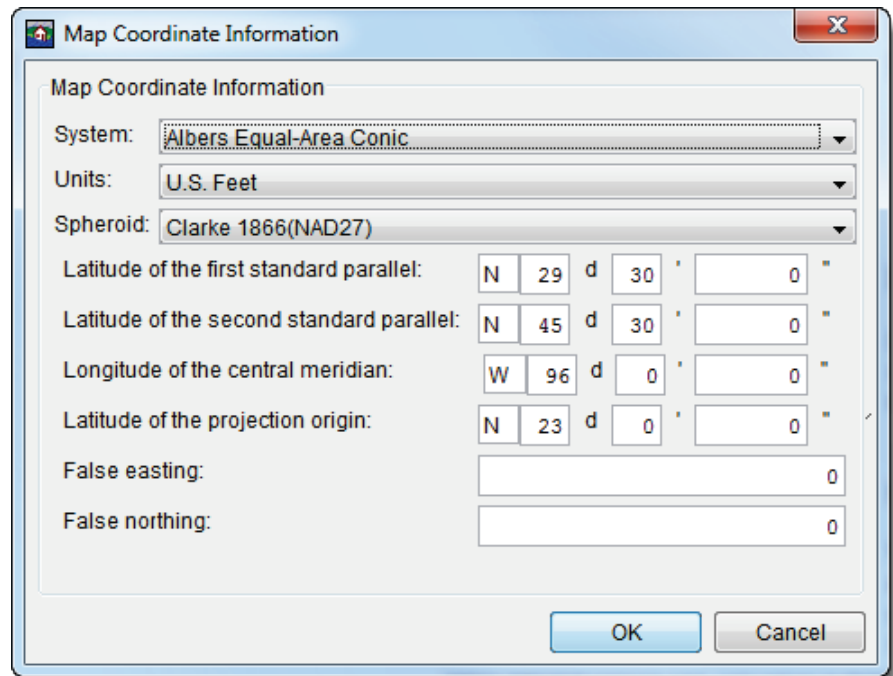
Figure D.8 Albers Equal-Area Conic Coordinate System.

6. The remaining items to be entered are the angular parameters that are required to fine-tune the projection. When specifying latitudes you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use the **SPACEBAR** to toggle between the entries.

7. Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Albers Equal-Area Conic* should be displayed.

D.2.6   Lambert Conformal Conic Coordinate System

The Lambert Conformal Conic projection is used extensively for mapping areas of the world with predominantly east-west orientation. Similar to the Albers Equal-Conic projection, however the projection is not done in an equal-area.

To set parameters for the Lambert Conformal Conic Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).

2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.9).

3. From the **System** list, select **Lambert Conformal Conic**.

4. From the **Units** list, select the units for the Lambert Conformal Conic coordinate system.

5. From the **Spheroid** list, select a spheroid type for the Lambert Conformal Conic coordinate system.
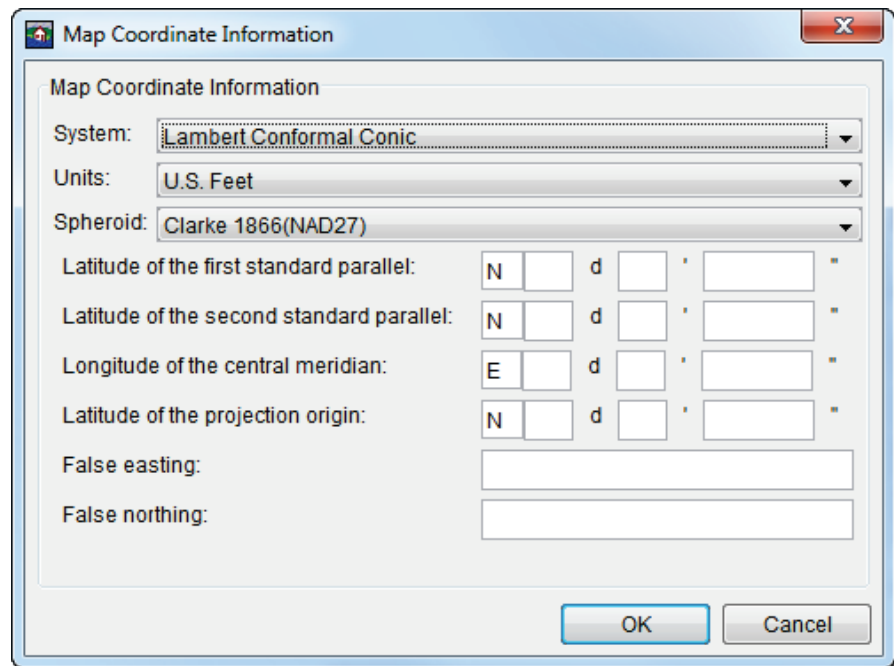
Figure D.9 Lambert Conformal Conic Coordinate System.

6.   When specifying latitudes you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use the **SPACEBAR** to toggle between the entries.

7.   Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Lambert Conformal Conic* should be displayed.

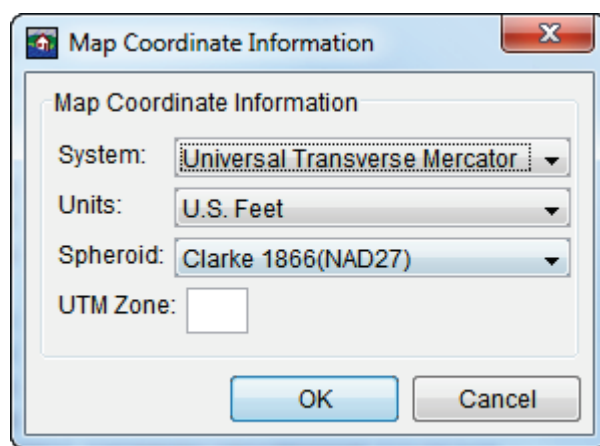## D.2.7   Transverse Mercator Coordinate System

In the Transverse Mercator projection, a sphere is projected onto a cylinder tangent to a central meridian. It is similar to the Lambert Conformal Conic project, but used to portray large areas in a north-south orientation. Many national grid systems are based on the Transverse Mercator projection.

To set parameters for the Transverse Mercator Coordinate System:

1.   On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).

2.   The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.10).

3.   From the **System** list, select **Transverse Mercator**.

4.   From the **Units** list, select the units for the Transverse Mercator coordinate system.
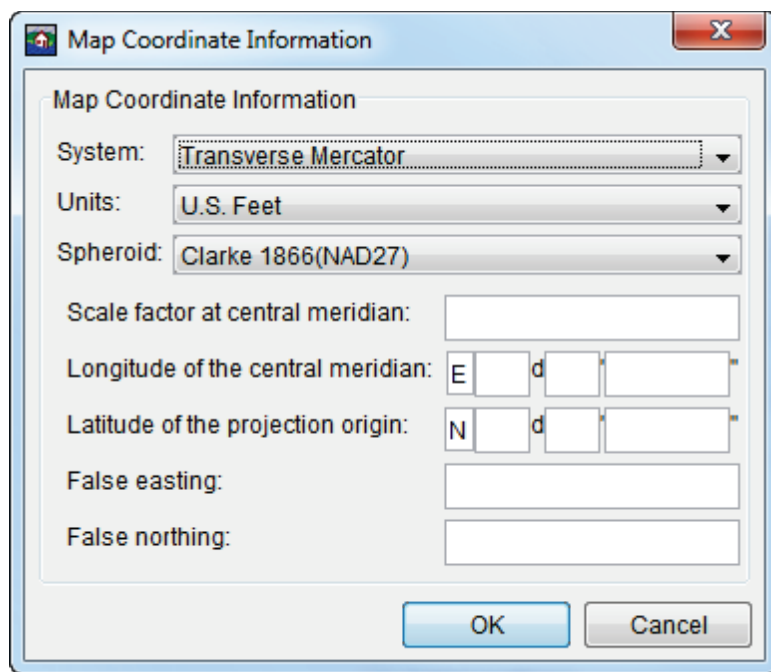
Figure D.10 Transverse Mercator Coordinate System.

5.   From the **Spheroid** list, select a spheroid type for the Transverse Mercator coordinate system.

6.   Enter a scaling factor for central meridian, in the **Scale factor at central meridian** box.

7.   When specifying latitudes you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use the **SPACEBAR** to toggle between the entries.

8.   Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Transverse Mercator* should be displayed.
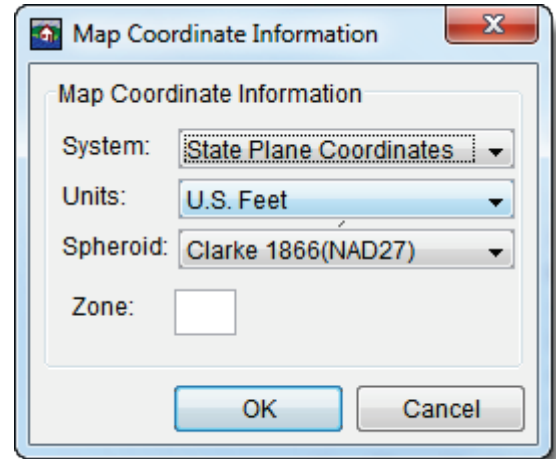
D.2.8   Albers Equal-Area Conic (SHG) Coordinate System

The standard hydrologic grid (SHG) is a variable-resolution square-celled map grid defined for the conterminous United States, based on the Albers Equal-Area Conic projection system. The parameters are set to match USACE map coordinates for SHG.

To view the parameters for the **Albers Equal-Area Conic (SHG) Coordinate System**:

1.   On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).

2.   The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.11).
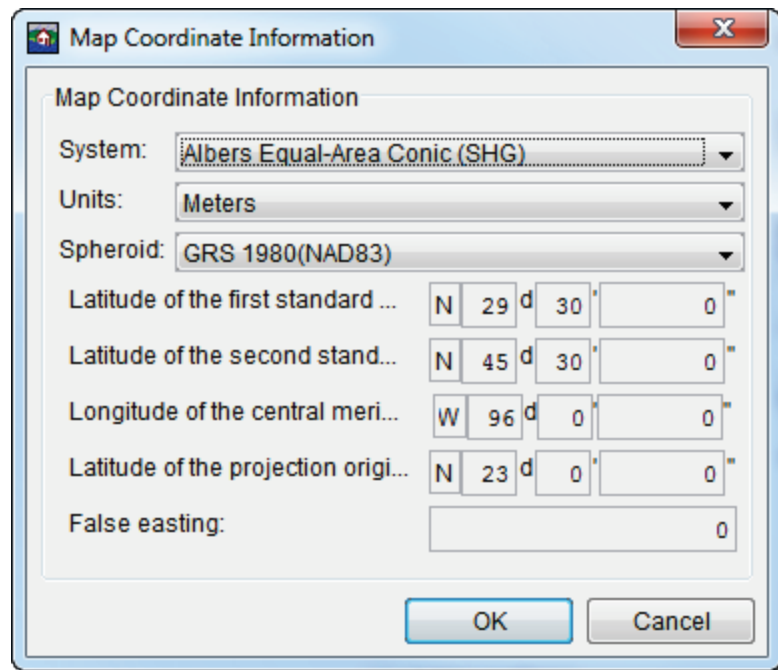
Figure D.11 Albers Equal-Area Conic (SHG) Coordinate System.

3. From the **System** list, select **Albers Equal-Area Conic (SHG)**.
4. From the **Units** list, select the units for the Albers Equal-Area Conic (SHG) coordinate system.
5. From the **Spheroid** list, select a spheroid type for the Albers Equal-Area Conic (SHG) coordinate system.
6. The remaining parameters are informational and are not editable.
7. Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Albers Equal-Area Conic (SHG)* should be displayed.

D.2.9   Polar Stereographic (HRAP) Coordinate System

The Hydrologic Rainfall Analysis Project (HRAP) projection system, a type of polar stereographic coordinate system, is used for navigation in polar regions. The parameters are set to match NWS map coordinates used for radar grids.

To view the parameters for the Polar Stereographic (HRAP) Coordinate System:

1. On the **Maps** menu, select **Default Map Properties**. The **Map Default Properties** dialog box will open (see Figure D.2).
2. The **Coordinate System** field displays the name of the selected coordinate system (X-Y is default). Click **Edit**. The **Map Coordinate Information** dialog box will open (Figure D.12).
3. From the **System** list, select **Polar Stereographic (HRAP)**.
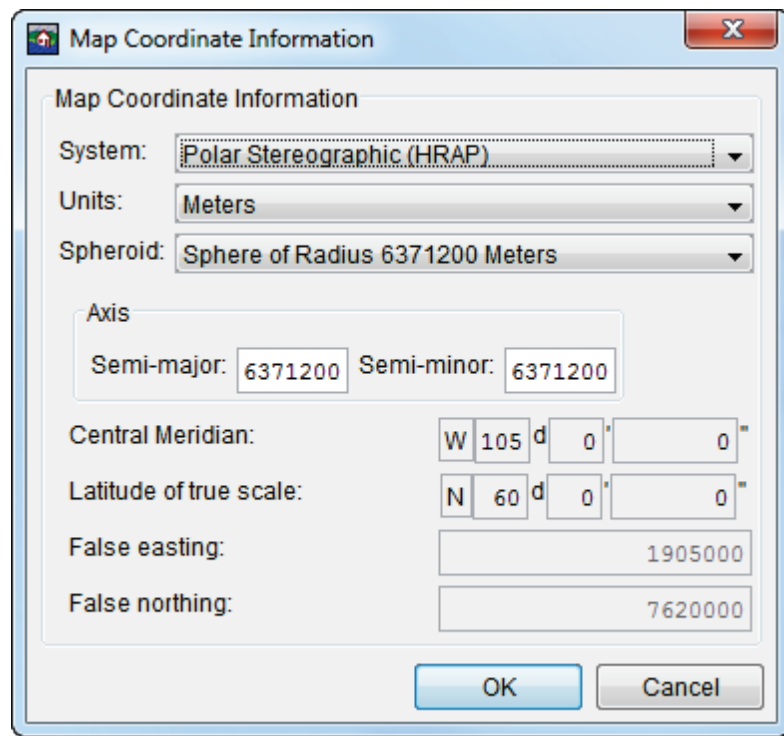
Figure D.12 Polar Stereographic (HRAP) Coordinate System.

4. From the **Units** list, select the units for the Polar Stereographic (HRAP) coordinate system.

5. From the **Spheroid** list, select a spheroid type for the Polar Stereographic (HRAP) coordinate system.

6. The values entered in the **Semi-major** and **Semi-minor** boxes reflect the length of the axes of an ellipse and will change according to spheroid selection. In Figure D.12, *Sphere of Radius 6371200* is selected as the spheroid, and the lengths of the axes have been automatically entered to reflect a sphere of radius 6371200 feet. These values are for information only and not user-editable.

7. The remaining parameters are informational and are not editable.

8. Click **OK**. The **Map Coordinate Information** dialog box will close, and you will be returned to the **Display Coordinate Information** dialog box. In the **Coordinate System** box, *Polar Stereographic (HRAP)* should be displayed.

Appendix E

Layer Display Properties

There are many map layers supported by FIA. The different map layers are described in Chapter 3, Section 3.2.1 (page 3-1) of this manual. Users can edit several display properties for each of these layers, and the different editor dialog boxes are layer-specific to polygon shapefiles, line shapefiles, and gridded data. This chapter will outline the functions of each of the editors for the different layers in FIA.

To access the map layer editors, beneath the **Content Pane**, select the **Maps Tab**. The **Content Pane** will display all of the study elements that are represented in the **Map Window**. Select the map layer you would like to edit and right-click. From the shortcut menu click **Properties**. An editor specific to the type of map layer you select will open.
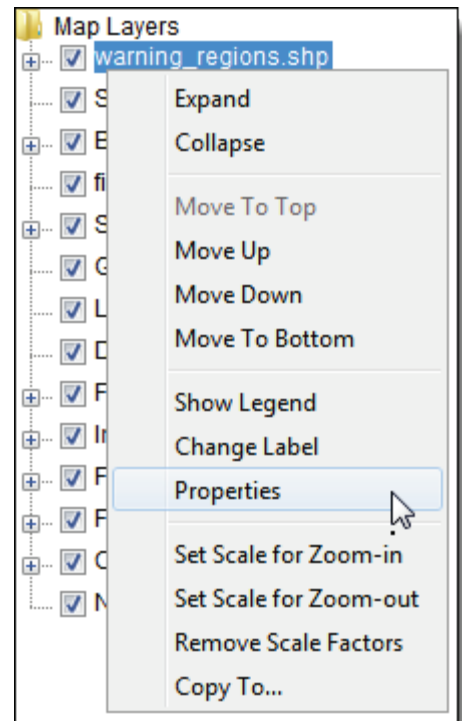
Figure E.1 Map layer shortcut menu.

E.1   Polygon Shapefile Display Properties

This layer type is the native data structure for the ArcView® GIS program. Shapefiles store nontopolgical geometry and attribute information for the spatial features of a data set. There are three types of shapefiles: polygon, line, and point. Each of these have a different set of display options.

User-editable polygon shapefile display properties include **Fill**, **Borders**, and **Labels**.

To edit polygon shapefile fill properties:

1. Starting in the Content Pane with the Maps Tab selected, select and right-click the name of the shapefile (**.shp*) you wish to edit.
2. From the shortcut menu, select **Properties** (Figure E.1). The **Edit Polygon Properties** dialog box will appear (Figure E.2).
3. Under the **Fill** tab, check the **Display Fill** box to fill the selected shapefile.
4. By default, FIA will draw fill features using **One Fill** for the selected shapefile. To set fill properties for a single fill:
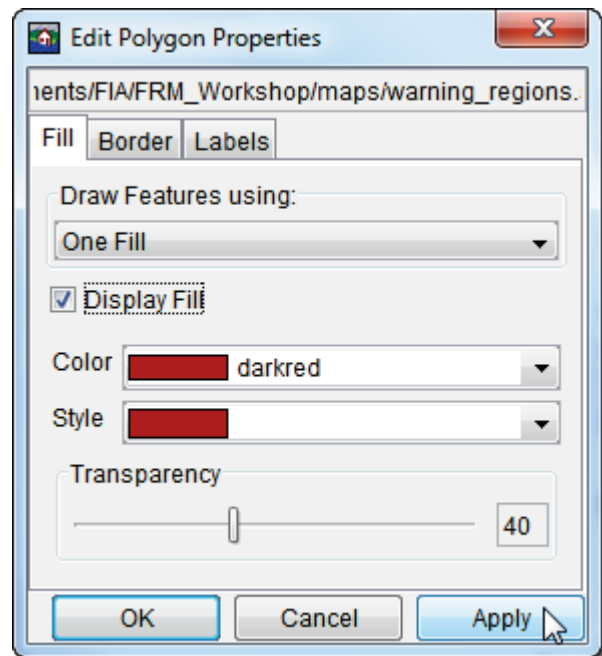
    Figure E.2 Edit Polygon Properties dialog box

    a. From the **Color** list, select the fill color. In the example in Figure E.2, dark red has been selected.
    b. From the **Style,** select the fill style (e.g., solid color, diagonal lines, etc.).
    b. Use the slider to select the level of fill transparency.
5. Users can also establish separate fill colors by shapefile attribute:
    a. From the **Draw Features using** dropdown list, select **Attribute Values**.
    b. From the **Field for values** list, select the name of the attribute you want to establish separate fills for. In Figure
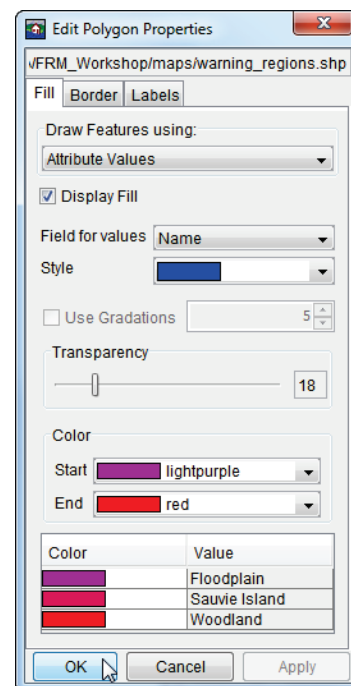
    Figure E.3 Fill polygon by shapefile attribute

> E.3, Name has been selected for the shapefile
> *warning_regions.shp*.
> c. From the **Style** dropdown list, select the desired fill style.
> d. Use the slider to select the level of fill transparency.
> e. Select the start and end of the color range for your
>   shapefile. FIA will fill in the colors in between. In Figure
>   E.4, the shapefile has been filled according to the
>   properties entered in Figure E.3.
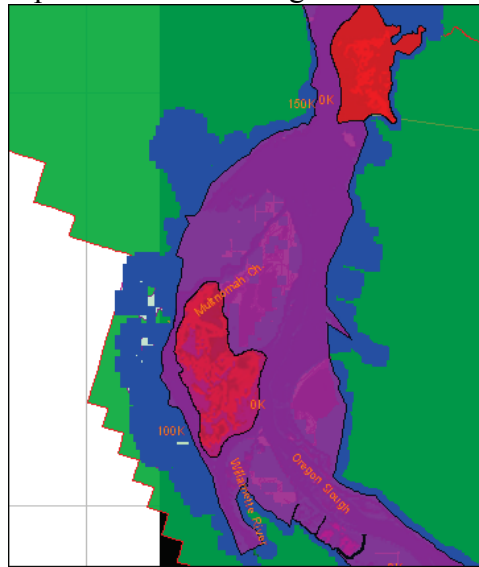
Figure E.4 Shapefile polygon filled by attribute

6. Once you have edited fill properties as desired, click **Apply** to
   apply the display properties to your layer. When finished editing,
   click **OK**.

To edit shapefile border properties:

1. Starting in the Content
   Pane with the Maps Tab
   selected, select and right-
   click the name of the
   shapefile (**.shp*) you wish
   to edit.
2. From the shortcut menu,
   select **Properties** (see
   Figure E.1). The **Edit
   Polygon Properties** dialog
   box will appear.
3. Select **Border** tab and
   check the **Display Border**
   box to create a border (see Figure E.5).
4. Edit the border color, style (e.g. solid, dashed, etc.), and weight as
   desired.
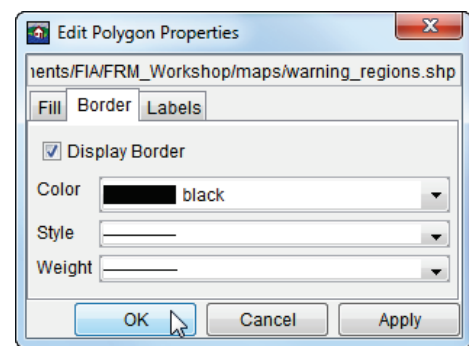
**Figure E.5 The Border tab of the
polygon properties editor**

5. Click **Apply** to display your changes in the **Map Window** and keep editing.
6. When finished editing, click **OK** to close the dialog.

To edit labels for shapefile attributes:

1. Starting in the Content Pane with the Maps Tab selected, select and right-click the name of the shapefile (**.shp*) you wish to edit.
2. From the shortcut menu, select **Properties** (see Figure E.1). The **Edit Polygon Properties** dialog box will appear.
3. Select the **Labels** tab (Figure E.6).

Figure E.6 Edit Polygon Properties dialog box—Labels.

4. From the Label features using dropdown list, select the shapefile attribute you wish to display labels for. In the example in , *Name* has been selected for the *warning_regions* shapefile to display the names of the different warning regions.
5. Edit the font style as desired and move the slider to select font size. You can also toggle placement and rotation of the labels.
6. Click **Apply** to display your changes in the **Map Window** and keep editing.
7. When finished editing, click **OK** to close the dialog.

E.2   Line Shapefile Display Properties

Line shapefiles are usually uploaded to studies as centerlines that represent a stream system for a study (see Chapter 5, Section 5.1.9, page 5-18).

User-editable line shapefile display properties include **Style** and **Labels**.

To edit line shapefile style properties:

1. Starting in the Content Pane with the Maps Tab selected, select and right-click the name of the line shapefile (e.g. *CRT_Wkshp_Centerline.shp*) you wish to edit.
2. From the shortcut menu, select **Properties** (see Figure E.1). The **Edit Line Properties** dialog box will appear (Figure E.2).
3. From the **Draw Features using** dropdown list, **One Style** is the default. This option will apply style properties to all of your shapefile.

    a. From the **Color** list, select the display color of the line shapefile.

    b. From **Style**, select the line style of the shapefile from the dropdown list.

    c. From **Weight**, select the desired weight (thickness) of the line.

4. Users can also **Draw Features using Shapefile Attribute**:

    a. From the **Draw Features using** dropdown list, select **Shapefile Attribute**.

    b. Additional display property options will appear (Figure E.8).

    c. From the **Field for values** list, select the shapefile attribute by which you wish to display your shapefile. In Figure E.8, *RiverCode* has been selected to display the names of the rivers represented in the line shapefile.

    d. Select the desired line style from the **Style** list**.**

    e. Select desired line weight (thickness) from the **Weight** list.

    f. Select a **Start** color and an **End** color. FIA will automatically fill in a gradation in between the two colors.

5. When you have made the desired changes, click **Apply** to display changes in the **Map Window**. When finished editing, click **OK** to close the dialog box.

Figure E.7 Edit Line Properties dialog box.
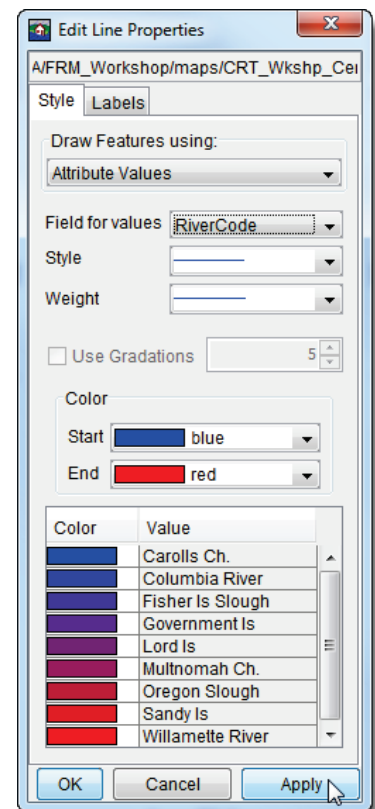
Figure E.8 Draw line features using shapefile attribute.

Users can display labels for attributes of line shapefiles and edit the way they display in the **Map Window**.

To edit label properties for line shapefiles:

1. Starting in the Content Pane with the Maps Tab selected, select and right-click the name of the shapefile (**.shp*) you wish to edit.
2. From the shortcut menu, select **Properties** (see Figure E.1). The **Edit Line Properties** dialog box will appear.
3. Select the **Labels** tab (Figure E.9).
4. From the **Label features using** dropdown list, select the shapefile attribute you wish to display labels for. In the example in Figure E.9, *RiverCode* has been selected for the *FRM_Wkshp_Centerline* shapefile to display the names of each river in the stream system.
5. Edit the font style as desired and move the slider to select font size. You can also toggle placement and rotation of the labels.
6. Click **Apply** to display your changes in the **Map Window** and keep editing.
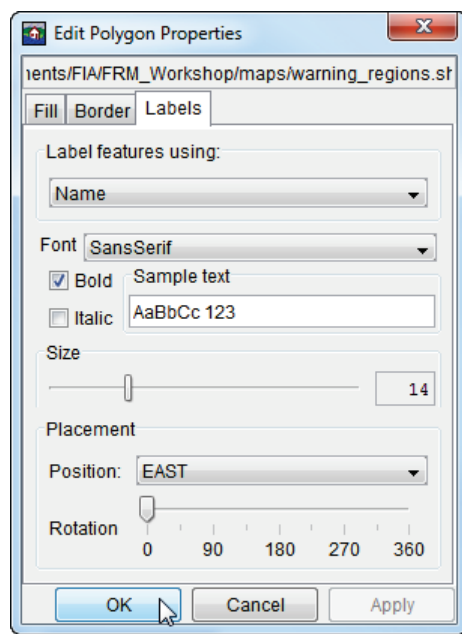7. When finished editing, click **OK** to close the dialog.
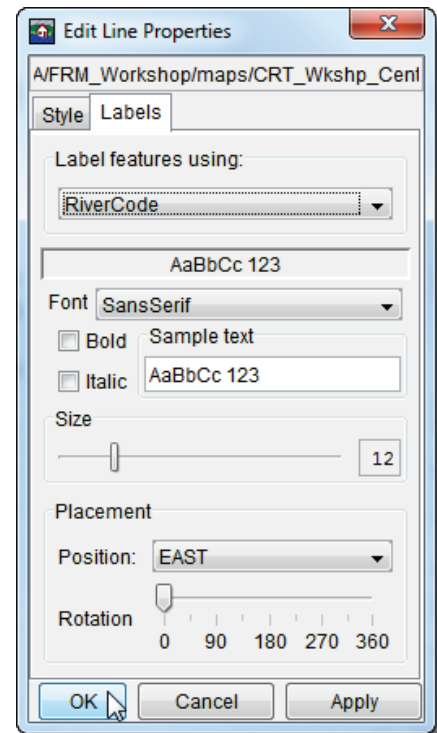
Figure E.9 Edit Line Properties dialog box—Labels.

E.3   Gridded Data Display Properties

For gridded data (.smrmp, .flt), users can adjust display properties for color by elevation as well as toggle at which scales the layer will be displayed in the map window.

To adjust display properties for data grids:

1. Starting in the **Content Pane** with the **Maps Tab** selected, select and right-click the name of the grid (**.smrmp*, **.flt*) you wish to edit.
2. From the shortcut menu, select **Properties** (see Figure E.1). The **Elevation Options** dialog box will appear (Figure E.10).
3. The dialog box will be on the **Properties** tab by default.
4. From the list box, there are available color contour schemes – **Aspect Shading, Grayscale, Linear, Precipitation, Red-Green-Blue,** and **Terrain** (default).
5. The **Draw Edges** check box is specifically for ASCII NetTin files. If selected, the edges of the triangles that make up an ASCII NetTin file will be drawn.
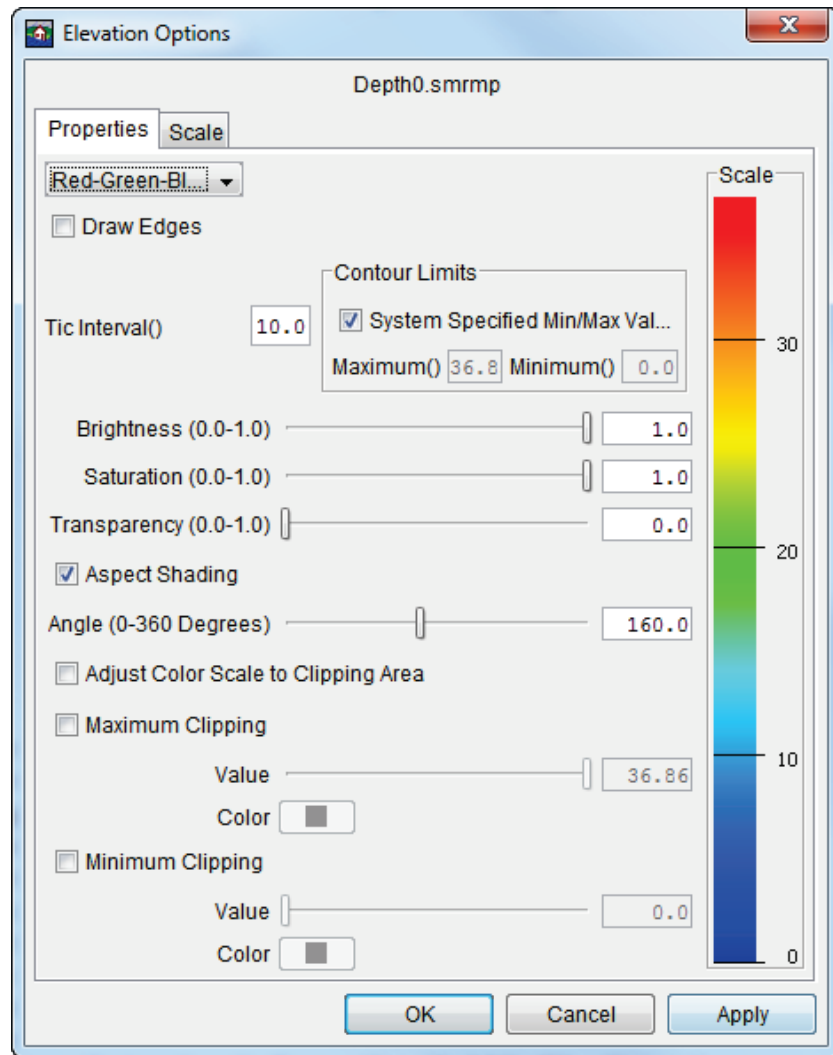
Figure E.10 Elevation Options dialog box

6. Set the contour tic interval by entering a value in the **Tic Interval** box. The program sets the maximum and minimum limits of the contours automatically. To set your own maximum and minimum limits, clear the **System Specified Min/Max Values** box and enter new values in the **Maximum** and **Minimum** boxes.

7. Adjust the brightness (amount of white), saturation (amount of black), and transparency (level of opacity or alpha) for the colors using the **Brightness**, **Saturation**, and **Transparency** slider bars or boxes.

8. By default the **Aspect Shading** option is on. Aspect shading displays the map layer in relief by placing an imaginary light source above the map and shading the contours. You can adjust the angle of the light source by using the **Angle** slider bar or by entering a value in the box. If you do not want aspect shading, clear the box labeled **Aspect Shading**.

9. Clipping allows users to highlight an area using user-definable color choices and values. For maximum clipping, fill the contour with the

clip color from the maximum value of the map to the value entered for **Maximum Clipping**. For example, maximum clipping could be used to highlight smog levels, cloud cover, or snow level at a particular elevation. By default, maximum clipping is turned off. To enable, click the **Maximum Clipping** check box, then choose a color by clicking the **Color** box, and either enter a maximum clipping value by using the slider bar or enter a value in the box.

10. Minimum clipping will fill the contour with the clip color from the minimum value of the map to the value entered for Minimum Clipping. For example, minimum clipping could be used to see where a water level would be if it flooded to a particular elevation. By default, minimum clipping is turned off. To enable, click the **Minimum Clipping** check box. Then choose a color by clicking the **Color** box, and either enter a minimum clipping value by using the slider bar or by entering a value in the box.

11. Click **Apply** to display your changes in the **Map Window** and keep editing.

12. When finished editing, click **OK** to close the dialog.

Users may also set the zoom scales at which they would like layers to display in the **Map Window**. For example, a user might not want to see their counties shapefile below a particular scale. To toggle these settings:

1. Starting in the **Content Pane** with the **Maps Tab** selected, select and right-click the name of the grid (*.smrmp*, *.flt*) you wish to edit.

2. From the shortcut menu, select **Properties** (see Figure E.1). The **Elevation Options** dialog box will appear.

3. Select the **Scale** tab (Figure E.11).
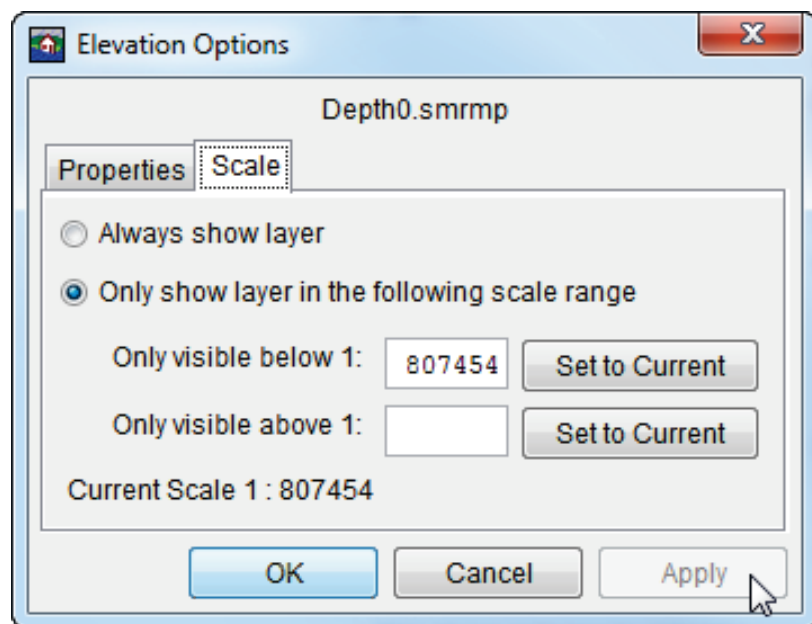
Figure E.11 The Scale tab of the elevation options

4. Select **Always Show Layer** to display the layer in the **Map Window** at all scales.
5. To make the layer visible only below a certain scale level, zoom to the desired scale level and select **Set to Current**. The layer will now appear only when the zoom scale is below this level.
6. To make the layer visible only above a certain scale level, zoom to the desired scale level and select **Set to Current**. The layer will now appear only when the zoom scale is higher than this level.
7. Click **Apply** to display your changes in the **Map Window** and keep editing.
8. When finished editing, click **OK** to close the dialog.

Appendix F

Using the Color Chooser

The **Color Chooser** (Figure F.1) affords great flexibility when you need to select default colors for map layers, labels, and background colors in your watershed display. The **Color Chooser** has three tabs: **Swatches**, **HSB**, and **RGB**, offering three methods for choosing a color. For each method, the preview area allows you to see your choice before applying changes.
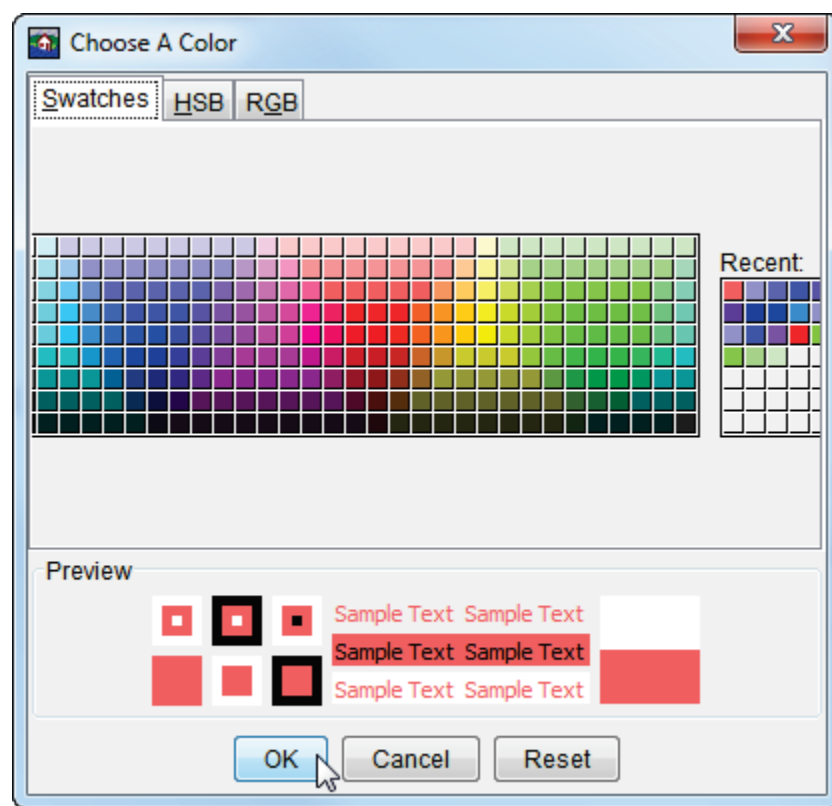
Figure F.1 Color Chooser—Swatches Worksheet.

F.1   Swatches Worksheet

The **Swatches** worksheet provides a palette of pre-defined colors. From the palette, select a color. Once you have selected a color, the **Recent** box displays that color, as also does the **Preview** box. Click **OK**, the **Color Chooser** will close. Depending on where you accessed the **Color Chooser**, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

F.2   HSB Worksheet

HSB is the Hue, Saturation, and Brightness color model, which allows you to set the hue, saturation, and brightness of the colors you wish to produce. From the **Color Chooser**, select the **HSB** tab; the **HSB Worksheet** (Figure F.2) becomes available.
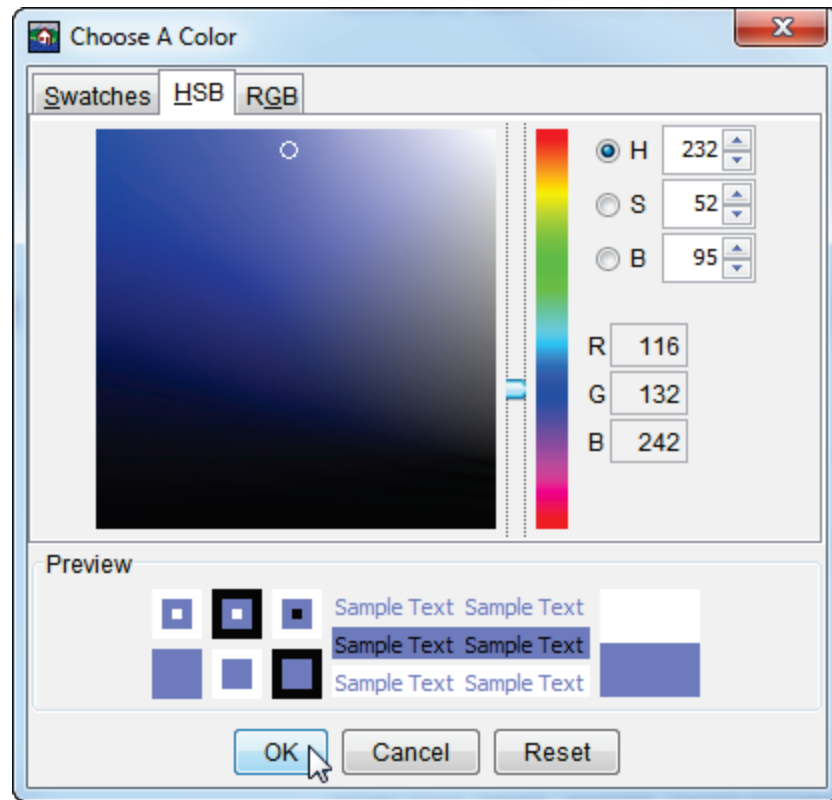
Figure F.2 Color Chooser—Hue, Saturation, and Brightness worksheet.

There are several ways to adjust the hue, saturation, and brightness of the colors. The slider bar in conjunction with the **H**, **S**, and **B** options will change the color, if you select *red* with the **H** option selected, then when you switch to the **S** and **B** options, you will be changing the saturation and brightness of *red*. Another way to affect the colors is in the **H**, **S**, and **B** boxes you can directly enter a value for hue, saturation, and brightness. The following rules apply to the values you can enter for each box:

H Box: Hue is measured in a circle from zero (0) to 359 degrees.
0 = red, 60 = yellow, 120 = green, 180 = cyan
240 = blue, 300 = magenta

S Box: Saturation is the amount of black, measured from zero (0) to one hundred (100) percent.

B Box: Brightness is the amount of white, measured from zero (0) to one hundred (100) percent.

A finally way to set a color on the HSB worksheet, is from the color palette, click and drag the little white circle. This will automatically change the hue, saturation, and brightness values, along with the red, green and blue color values. The red, green, and blue values in the **R**, **G**, and **B** boxes are not editable, and informational only.

Click **OK**, the **Color Chooser** will close. Depending on where you accessed the **Color Chooser**, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

F.3   RGB Worksheet

RGB is the Red, Green, and Blue color model, which allows you to set the red, green, and blue values of the colors you wish to produce. From the **Color Chooser**, select the **RGB** tab; the **RBB Worksheet** (Figure F.3) becomes available.

There are two ways to adjust the red, green, and blue value of colors. There is a slider bar for **Red**, **Green**, and **Blue**. As you change the slider bar for each one, in the **Preview** box, the color you are producing is shown. Another way to affect colors is in the **Red**, **Green**, and **Blue** boxes you can directly enter a value. Click **OK**, the **Color Chooser** will close. Depending on where you accessed the **Color Chooser**, you will have to click either **Apply** or **OK** for the color change to appear in the display area.
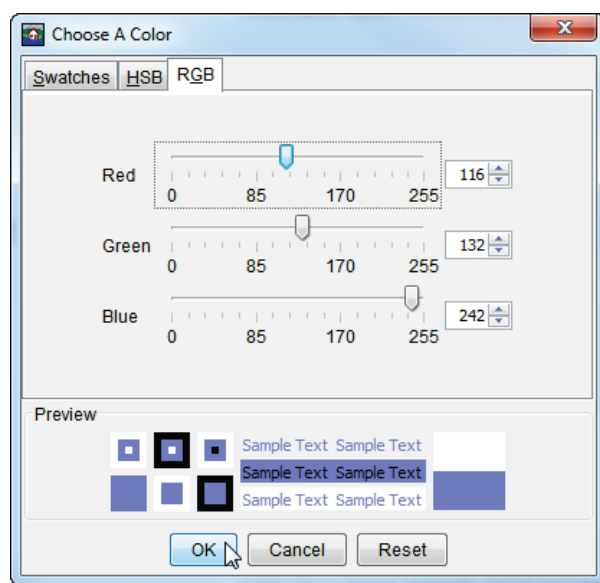
Figure F.3 Color Chooser—RGB worksheet.

Appendix G

Using HEC-DSSVue

HEC-DSSVue (Hydrologic Engineering Center Data Storage System Visual Utility Engine) is a graphical user interface program for viewing, editing, and manipulating data in HEC-DSS database files. With DSSVue, data may be plotted, tabulated, edited, and manipulated with over fifty mathematical functions. Along with these functions, DSSVue provides several utility functions, such as entering data sets into a database, renaming data set names, copying data sets to other DSS database files, and deleting data sets.

In FIA, you will use DSSVue mainly as a viewing tool for DSS data. This appendix will review some of the general functions in DSSVue. For further detail, see the HEC-DSSVue User's Manual.

G.1   HEC Data Storage System Concepts

HEC-DSS is a database system that was designed to efficiently store and retrieve scientific data that is typically sequential. Such data types include, but are not limited to, time series data, curve data, spatial-oriented gridded data, textual data (such as this manual), and others. The system was designed to make it easy for users and application programs to retrieve and store data.

The data are arranged into blocks. For example, one year of daily-interval data is stored in a single **record** in the DSS. Each record is given a unique name called a pathname. The **pathname** is formulated in a way that makes it easy to group related data together. Software from HEC has the ability to read from and write to DSS. This capability facilitates data exchange among applications. For example, you could use ResSim to generate the time series input for FIA.

G.2   Overview of HEC-DSSVue in FIA

From the **Tools** menu, choose **HEC-DSSVue**. The **DSSVue** main window will open. The main window and the different components are illustrated in Figure G.1.

The **Menu Bar** contains the menus for the main window of DSSVue.
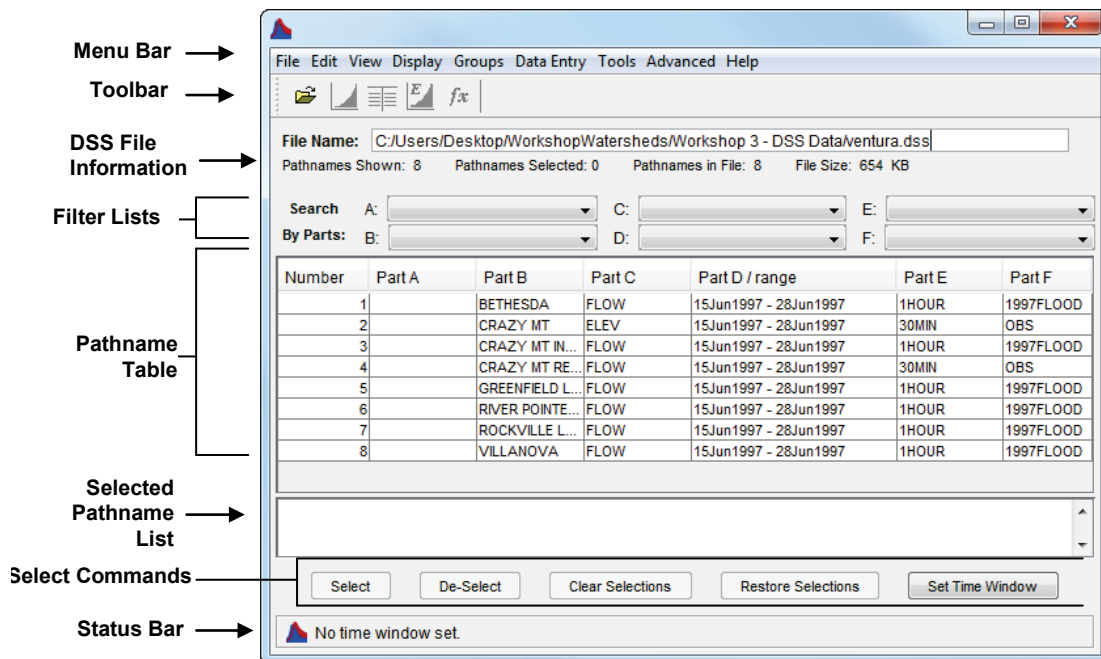
Figure G.1 HEC-DSSVue main window.

The **Toolbar** contains shortcut buttons to the most-used features of DSSVue:

   Open File Button – opens an **Open File Browser**, which allows you to select a DSS file.

   Plot Button – plots the data of the selected pathname(s).

   Tabulate Button – displays the data of the selected pathname(s) in a tabular form.

   Graphical Edit Button – provides a shortcut to an editor for data manipulation.

   Function Button – provides a shortcut to the mathematical function tools for editing DSS data.

The **DSS File Information** (see Figure G.1) box displays the filename of the DSS file, and statistics about the pathnames in the DSS file and the size of the DSS file.

The **Filter Lists** are used to sort DSS pathnames by parts. There is a filter list for each individual part, A through F. The **Pathname Table** displays the filtered pathnames contained in the DSS file. The pathnames are displayed by the individual parts. The **Selected Pathname List** displays the pathnames that you have selected from the **Pathname Table**.

The **Select Commands** are available after you have selected the pathnames you wish to view or manipulate. Available commands are **Select, De-Select, Clear Selections, Restore Selections**, and **Set Time Window**. These commands will be detailed later on in this appendix.

When you have entered time window information, the **Status Bar** will display the time window information. Users can set a time window to view data within a particular time range.

G.3   Opening a DSS File

There are several ways to open a DSS file from DSSVue. One way is to type in the location and filename of the DSS file in the **File Name** box of the **DSSVue** main window (see Figure G.1). After entering the correct information, press **ENTER**. The **Pathname Table** will display the pathnames of the selected DSS file, and the **DSS File Information** box will display statistics about the DSS file.

Alternatively, select **Open** from the **File** menu or click the **Open File** button 📂 from the **Toolbar**. Either method will open an **Open HEC-DSS File** browser (Figure G.2).

Locate the DSS file you wish to open, click **OK**. The **Open HEC-DSS File** browser will close,

Figure G.2 Open HEC-DSS File browser.

and the DSSVue main window will display the name of the DSS file and all of the pathnames associated with that DSS file will appear in the **Pathname Table** (see Figure G.1).
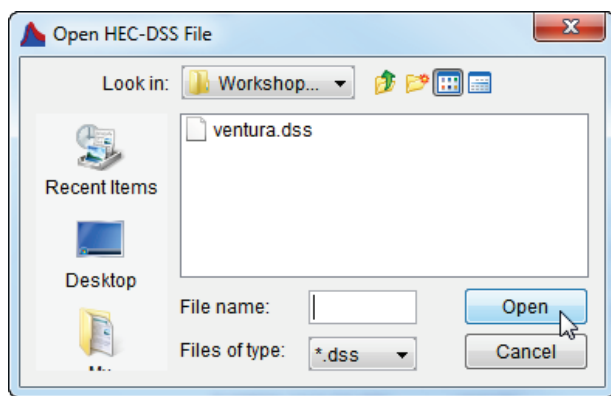
G.4   Set Time Window

To view data for a particular block of time, users can define a time window. Users can define a time window by specific starting and ending dates and times, by time relative to the current date, or by individual water years.

1. Click **Set Time Window**, the **Set Time Window** dialog box will open (Figure G.3). By default, **No Time Window** will be selected.

2. To enter specific start
    date and time:
    a. Select
        **Specific
        Time
        Window**.
    b. In the **Start
        Date** box
        type a date in
        the format
        ddmmmyyyy
        (i.e.,
        26Nov1993),
        and in the
        Start Time
        box type a
        time in the
        format hhmm
        (i.e., *0630*).
    c. Enter an **End
        Date** and
        Time in the
        provided
        fields.
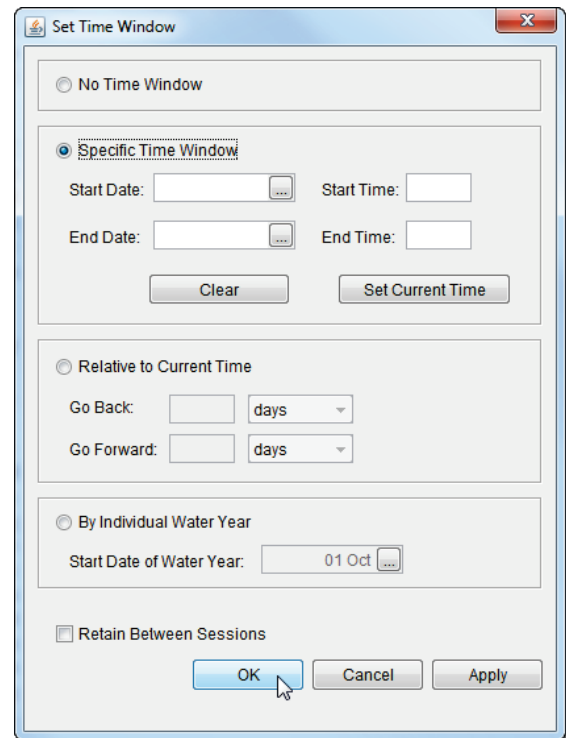
Figure G.3 Set Time Window dialog box.

    d. If you want to use the current date and time, click **Set
        Current Time**. All time window boxes will be filled, you
        will probably need to edit the ending data and time.
3. To set a time window relative to current time:
    a. Select **Relative to Current Time**.
    b. From the dropdown list, select time units (days, hours,
        months, or years).
    c. In the **Go Back** field, enter the number of
        days/hours/months/years you would like DSSVue to
        show prior to the current time.
    d. In the **Go Forward** field, enter the number of
        days/hours/months/years you would like DSSVue to
        display after the current time.
4. To set a time window by individual water year, in the **Start Date of
    Water Year** field select the date that represents the start of each
    water year.
5. If you would like to retain these time window settings for all data
    viewing in HEC-DSSVue, check **Retain Between Sessions**.
6. Click **OK**, the **Set Time Window** dialog box will close, the time
    window will be set, and the time window will be displayed in the
    Status Bar of the **DSSVue** main window (see Figure G.1).

G.5   DSS Pathnames

Data is stored in blocks, or records, within a file, and each record is identified by a unique name called a **pathname**. Each time that data is stored or retrieved from the file, its pathname must be given. The pathname is the key to the data's location within a database. A pathname is analyzed by DSS to determine an index number. This index determines where the data set is stored within the database. This design ensures that very few disk accesses are made to retrieve or store data sets. One data set is not directly related to another, so there is no need to update other areas of the database when a new data set is stored.

DSS references records (data sets) by a pathname. The pathname of a DSS record can be three hundred and ninety-one (391) characters or less, and is, by convention, separated into **six parts**. The parts can be sixty-four (64) characters or less and are delimited by slashes in the pathname. The pathname parts are designated **A** through **F**. **Table G-1** provides an example naming convention for regular-interval time series:

Table G.1 DSS Pathname Parts

A	project, river, or basin name
B	location identifier
C	data parameter (e.g., FLOW, PRECIP)
D	starting block data; format ddmmyyyy
E	time interval
F	additional user-defined descriptive information

G.5.1   Viewing of DSS Pathnames

The **Pathname Table** displays all of the pathnames by their different parts (see Figure G.1). Users can toggle display options from the **View** menu. Available commands are **Pathname List**, **Pathname Parts**, **Condensed Catalog**, **No Pathnames**, and **Unsorted List**. The **Pathname List** command (Figure G.4) displays the pathnames in the **Pathname Table**, while **No Pathnames** displays nothing.

Number	Pathname
1	//BETHESDA/FLOW/01JUN1997/1HOUR/1997FLOOD/
2	//CRAZY MT/ELEV/01JUN1997/30MIN/OBS/
3	//CRAZY MT INFLOW/FLOW/01JUN1997/1HOUR/1997FLOOD/
4	//CRAZY MT RELEASE/FLOW/01JUN1997/30MIN/OBS/
5	//GREENFIELD LOCAL/FLOW/01JUN1997/1HOUR/1997FLOOD/
6	//RIVER POINTE LOCAL/FLOW/01JUN1997/1HOUR/1997FLOOD/
7	//ROCKVILLE LOCAL/FLOW/01JUN1997/1HOUR/1997FLOOD/
8	//VILLANOVA/FLOW/01JUN1997/1HOUR/1997FLOOD/

Figure G.4 Pathname List.

G.5.2   Sorting DSS Pathnames by Parts

Users can also filter the display by pathname parts. Sorting the DSS pathnames by parts is the command **Search pathnames by part** (default). However, another way to sort the pathnames is available from the **View** menu, choose **Search pathnames by string**. The details for this method are described in the HEC-DSSVue User's Manual.

To filter by pathname parts, from the **Filter Lists** (Figure G.5) decide which pathname parts will be used to sort the pathname list. There is a **Filter List** for each individual pathname part.

In the example in Figure G.5, the **D Part** *01JUN1997* has been selected to show all pathnames with that **D Part**.

Number	Part A	Part B	Part C	Part D	Part E	Part F
1		BETHESDA	FLOW	01JUN1997	1HOUR	1997FLOOD
2		CRAZY MT	ELEV	01JUN1997	30MIN	OBS
3		CRAZY MT IN...	FLOW	01JUN1997	1HOUR	1997FLOOD
4		CRAZY MT RE...	FLOW	01JUN1997	30MIN	OBS
5		GREENFIELD L...	FLOW	01JUN1997	1HOUR	1997FLOOD
6		RIVER POINTE...	FLOW	01JUN1997	1HOUR	1997FLOOD
7		ROCKVILLE L...	FLOW	01JUN1997	1HOUR	1997FLOOD
8		VILLANOVA	FLOW	01JUN1997	1HOUR	1997FLOOD

Search By Parts: A: ____ ▾ C: ____ ▾ E: ____ ▾ B: ____ ▾ D: 01JUN1997 ▾ F: ____ ▾

Figure G.5 Sort DSS pathnames by parts.

G.5.3   Selecting Pathnames

Once the **Pathname Table** is sorted as desired, users can select which pathnames they want to view or manipulate. To select a pathname, double-click on a pathname in the **Pathname Table** and the selected pathname will appear in the **Selected Pathname List** (see Figure G.1). You can also click on a pathname in the **Pathname Table**, and then click the **Select** button.

To select multiple consecutive pathnames, in the **Pathname Table**, click on the first pathname in the consecutive series. Find the last pathname in the series and click on it while holding down the **SHIFT** key. The block of pathnames will be highlighted. Click **Select**, and all the pathnames within that block will be added to the **Selected Pathname List**.

Users can also select multiple pathnames by selecting them while holding down the **CTRL** key. Once all of the appropriate pathnames are selected,

release the **CTRL** key and click **Select**. All the pathnames you have selected will be added to the **Selected Pathname List**.

You can select all of the pathnames displayed in the **Pathname Table**. From the **Edit** menu, choose **Select All**. All of the pathnames will now appear in the **Selected Pathname List**.

G.5.4   Clearing Selected Pathnames

To clear a pathname from the **Selected Pathname List**, select a pathname and click **De-Select**. The pathname will be cleared from the **Selected Pathname List**. To clear all pathnames from the **Selected Pathname List**, click **Clear Selections**, and the **Selected Pathname Box** will be empty.

You can also clear groups of pathnames by using the same concepts described in the previous section selecting pathnames by using the **SHIFT** and **CTRL** keys.

Once pathnames have been cleared from the **Selected Pathname Box**, the pathnames can be restored to the **Selected Pathname Box** by clicking **Restore Selections**.

G.6   Visualizing Data

Once pathname(s) are selected, users can visualize them in graphical form (plots) or in tabular form. Further details on plots and tables are discussed in the HEC-DSSVue User's Manual.

G.6.1   Plots

To view a plot of the pathnames in the **Selected Pathname List** (see Figure G.1), either click the **Plot** button from the **Toolbar** or choose **Plot** from the **Display** menu. Both of these options will open the **Plot Window** (Figure G.6).

The parameters for the plots are labeled and color-coded. Date ranges are specified along one axis, and the units of measure are specified along the other axis. You can resize the **Plot Window** by clicking on the corner of the window and dragging it to the desired size.

You can also zoom in and out on areas of the plot using the **Zoom Tool**.

G.6.2   Tables

To view a table of the pathnames in the **Selected Pathname List** (see Figure G.7), either click the **Tabulate** button ⊞ from the **Toolbar** or
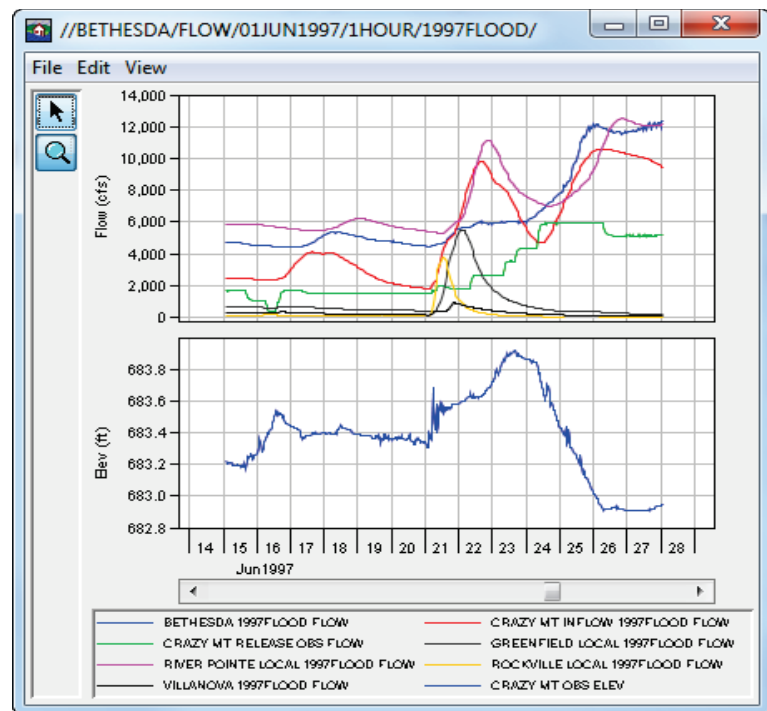
Figure G.6 Plot Window.

choose **Tabulate** from the **Display** menu. Either way will open the **Tabulate Window** (Figure G.7).

Figure G.7 Tabulate Window.

The **Tabulate Window** displays the ordinate (starting from the start date/time), the date and time stamp, and the values for the selected data sets. From the **Tabulate Window**, users have several options for displaying the tabular data (e.g. displaying commas in the number formatting, setting the precision of decimal places for your data).

G.7   DSS Records

HEC-DSS uses a block of sequential data as the basic unit of storage. Each block contains a series of values of a single variable over a time span appropriate for most applications. The basic purpose of DSS is the organization of data into records of continuous, applications-related elements, as opposed to individually addressable data items. This approach is more efficient for scientific applications than that of a conventional database system because it avoids the processing and storage overhead required for assembling an equivalent record from a conventional system. Data is stored in blocks, or records, within a file, and each record is identified by a unique name called a **pathname**.

G.7.1   Renaming Records

To change the pathname parts of a pathname:

1. You must have selected one or more pathnames for this command to be available (see Section G.5.3, page G-6 for pathname selection).
2. From the **Edit** menu, choose **Rename Records**. The **Rename Records to:** dialog box (Figure G.8) will open.

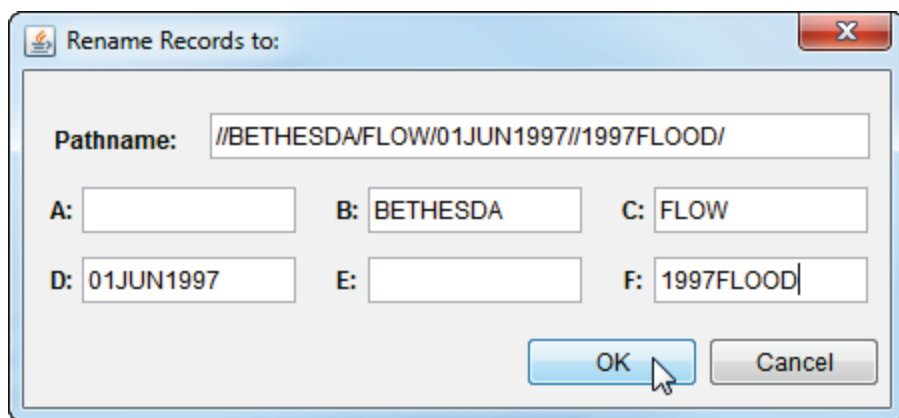**Figure G.8 Rename Records to dialog box.**

3. Each of the boxes (**A** - **F**) represents the different pathname parts. Users can change the parts by typing the new information in the appropriate box.
4. Click **OK**. The **Rename Records to:** dialog box will close and a message will appear displaying the number of records renamed. The selected pathname will no longer be displayed in the **Selected**

Pathname Box and the pathname will display in the **Pathname Table** (see Figure G.1) with the new pathname parts.

G.7.2   Duplicating Records

Users can duplicate records to make a copy of a record with an altered pathname part. To duplicate records:

1. You must have selected one or more pathnames (see Section G.5.3, page G-6) for this command to be available.
2. From the **Edit** menu, choose **Duplicate**. The **New pathname parts for duplicate records:** dialog box will open (similar to Figure G.8).
3. Each of the boxes (**A** - **F**) represents the different pathname parts. Users can enter new names for each of the parts in the fields.
4. Click **OK**, the **New pathname parts for duplicate records:** dialog box will close, a message will appear informing you of the number of records that were duplicated. A new pathname will be displayed in the **Pathname Table** (see Figure G.1).

G.8   Deleting Records

To delete records:

1. You must have selected one or more pathnames (see Section G.5.3, page G-6) for this command to be available.
2. A warning message dialog box appears asking if you really want to delete the listed pathnames.
3. To delete, click **Yes**. The warning message dialog box will close. A message will appear displaying the number of records deleted.
4. The deleted records will no longer appear in the **Pathname Table**.

G.9   Editing Data Associated with Records

If they have the appropriate permissions, users can edit the data associated with a pathname. To edit, from the **Edit** menu click **Tabular Edit**. Users can edit data directly, enter new time-series data or paired data, or use math functions to manipulate the data. All of these features are detailed in the HEC-DSSVue User's Manual.

Appendix H

Terms and Conditions for Use

Use of the program is governed by the terms and conditions for use. They limit what can be done with the program software, waive warranty, limit liability, and indemnify the developers and the United States government. The program cannot be used unless the terms and conditions for use are accepted; the full text is given below.

Terms and Conditions for Use

The United States Government, US Army Corps of Engineers, Hydrologic Engineering Center ("HEC") grants to the user the rights to install Flood Impact Analysis (HEC-FIA) "the Software" (either from a disk copy obtained from HEC, a distributor or another user or by downloading it from a network) and to use, copy and/or distribute copies of the Software to other users, subject to the following Terms and Conditions for Use:

All copies of the Software received or reproduced by or for user pursuant to the authority of this Terms and Conditions for Use will be and remain the property of HEC.

User may reproduce and distribute the Software provided that the recipient agrees to the Terms and Conditions for Use noted herein.

HEC is solely responsible for the content of the Software. The Software may not be modified, abridged, decompiled, disassembled, unobfuscated or reverse engineered. The user is solely responsible for the content, interactions, and effects of any and all amendments, if present, whether they be extension modules, language resource bundles, scripts or any other amendment.

The name "HEC-FIA" must not be used to endorse or promote products derived from the Software. Products derived from the Software may not be called "HEC-FIA" nor may any part of the "HEC-FIA" name appear within the name of derived products.

No part of this Terms and Conditions for Use may be modified, deleted or obliterated from the Software.

No part of the Software may be exported or re-exported in contravention of U.S. export laws or regulations.

Waiver of Warranty

THE UNITED STATES GOVERNMENT AND ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS PROVIDE HEC-FIA \"AS IS\", WITHOUT ANY WARRANTY OR CONDITION, EXPRESS, IMPLIED OR STATUTORY, AND SPECIFICALLY DISCLAIM ANY IMPLIED WARRANTIES OF TITLE, MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT. Depending on state law, the foregoing disclaimer may not apply to you, and you may also have other legal rights that vary from state to state.

Limitation of Liability

IN NO EVENT SHALL THE UNITED STATES GOVERNMENT AND ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS, BE LIABLE FOR LOST PROFITS OR ANY SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR IN CONNECTION WITH USE OF HEC-FIA REGARDLESS OF CAUSE, INCLUDING NEGLIGENCE.

THE UNITED STATES GOVERNMENT'S LIABILITY, AND THE LIABILITY OF ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS, TO YOU OR ANY THIRD PARTIES IN ANY CIRCUMSTANCE IS LIMITED TO THE REPLACEMENT OF CERTIFIED COPIES OF HEC-FIA WITH IDENTIFIED ERRORS CORRECTED. Depending on state law, the above limitation or exclusion may not apply to you.

Indemnity

As a voluntary user of HEC-FIA you agree to indemnify and hold the United States Government, and its agencies, officials, representatives, and employees, including its contractors and suppliers, harmless from any claim or demand, including reasonable attorneys' fees, made by any third party due to or arising out of your use of HEC-FIA or breach of this Agreement or your violation of any law or the rights of a third party.

Assent

By using this program you voluntarily accept these terms and conditions. If you do not agree to these terms and conditions, uninstall the program and return any program materials to HEC (if you downloaded the program and do not have disk media, please delete all copies, and cease using the program.)