FLO-2D©

Two-Dimensional Flood Routing Model

Workshop Lessons

Release 2009

FLO-2D Software, Inc.

FLO-2D©

Two-Dimensional Flood Routing Model

Workshop Lessons

Release 2009

FLO-2D Software, Inc.

FLO-2D©

Two-Dimensional Flood Routing Model

Workshop Lessons

Release 2009

FLO-2D Software, Inc.

FLO-2D Model Training Class

Presented to Flood Control District of Maricopa County
Phoenix, Arizona

Dates: June 27-28, 2013

Location: Maricopa County Public Work Operations Building Computer Training Room
2919 W Durango St, Phoenix Arizona 85009
(South side of MCDOT Traffic Operation Building; see attached map)

Instructors: Jimmy O'Brien, Ph.D., P.E.
Karen O'Brien, Technical Support and Instructor

WHO SHOULD ATTEND: This training class is designed for new FLO-2D users (hydraulic and hydrologic engineers and floodplain managers) and previous users who need a refresher class. The free FLO-2D Basic model available at the website will be used in this training.

Primarily 'hands-on' computer session times are highlighted in blue.

Thursday June 27, 2013
Getting Started

8:00 – 8:15	*Check-in, introductions and review agenda.*
8:15 – 9:00	*Overview of the FLO-2D Modeling system and Porting Data Files.*
9:00 – 10:00	*Grid Developer System - Getting Started*
10:00 – 10:15	***Break***
10:15 – 11:00	*Hands-on session 1: Lesson 1. Using the GDS to import and edit terrain elevation data, filter elevation point data, establish a grid system, work with aerial images, setup hydrographs and run the FLO-2D model.*
11:00 – 12:00	*Discussion of model theory, routing algorithms and stability criteria.*
12:00 – 13:00	***Lunch***
13:00 – 13:30	Review FLO-2D *.DAT and *.OUT files. *Introduction to floodplain attributes.*
13:30 – 14:15	*Hands-on session 2: Lesson 2. Floodplain attributes. Edit model components and layer attributes using shape files.*
14:15 – 15:00	*Hydrology: rainfall and inflow hydrographs. Using Green Ampt and SCS-CN infiltration.*
15:00 – 15:15	***Break***
15:15 – 15:45	*Hands-on session 3. Rainfall Lesson 7. Enter and edit rainfall data. (Demo)*
15:45 – 16:15	Introduction to Urban modeling.
16:15 – 17:00	*Hands-on session 4: Lesson 11. Urbanized floodplain details. Set up streets and buildings.*

Friday, June 28, 2013
Channel Flood Routing, Mapping and Troubleshooting

8:00 – 8:20	*Channel flood routing overview. Channel/floodplain flow exchange.*
8:20 – 8:45	*Overview of GDS river channel tools. Introduction to Lessons 3, 5 and 8.*
8:45 – 9:30	*Hands-on session 5: Lessons 3, 4 and 5. Using GDS to create a simple rectangular channel. Interpolating the channel cross sections and slope and editing the bank elements in PROFILES.*
9:30 – 10:30	*Channel routing problems: What to look for? NOFLOCs, channel profiles, n-values, surging.*
10:30 – 10:45	***Break***
10:45 – 11:30	*Hands-on session 6: Lesson 8. Import a HEC-RAS project and create a natural channel in the GDS.*
11:30 – 12:00	*Using hydraulic structures in the urban environment: weirs, bridges and culverts for rivers and floodplains*
12:00 – 13:00	***Lunch***
13:00 – 13:30	*Mapper overview. Creating high resolution flood delineation maps.*
13:30 – 14:15	*Hands-on session 7: Mapper (Demo Session) Display flow depths, velocities, and water surface elevation maps. Create shaded contour maps and import aerial photos.*
14:15 – 14:30	*Troubleshooting overview, tools and methods for finding data errors.*
14:30 – 15:00	*Hands-on session 8: Troubleshoot a project.*
15:00 – 15:15	***Break***
15:15 - 16:00	*Optimization: Is the flood simulation running ok? Solving common problems: Volume conservation, numerical surging, sticky grid elements, and limiting Froude numbers.*
16:00 – 17:00	*Hands-on session 9: Create a project from scratch.*

FLO-2D Model Training Class

Presented to Flood Control District of Maricopa County
Phoenix, Arizona

Dates: June 27-28, 2013

Location: Maricopa County Public Work Operations Building Computer Training Room
2919 W Durango St, Phoenix Arizona 85009
(South side of MCDOT Traffic Operation Building; see attached map)

Instructors: Jimmy O'Brien, Ph.D., P.E.
Karen O'Brien, Technical Support and Instructor

WHO SHOULD ATTEND: This training class is designed for new FLO-2D users (hydraulic and hydrologic engineers and floodplain managers) and previous users who need a refresher class. The free FLO-2D Basic model available at the website will be used in this training.

Primarily 'hands-on' computer session times are highlighted in blue.

Thursday, June 27, 2013
Getting Started

8:00 – 8:15 *Check-in, introductions and review agenda.*

8:15 – 9:00 *Overview of the FLO-2D Modeling system and Porting Data Files.*

9:00 – 10:00 *Grid Developer System - Getting Started*

10:00 – 10:15 ***Break***

10:15 – 11:00 *Hands-on session 1: Lesson 1. Using the GDS to import and edit terrain elevation data, filter elevation point data, establish a grid system, work with aerial images, setup hydrographs and run the FLO-2D model.*

11:00 – 12:00 *Discussion of routing algorithms and stability criteria.*

12:00 – 13:00 ***Lunch***

13:00 – 13:15 *Introduction to Lesson 2.*

13:15 – 14:15 *Hands-on session 2: Lesson 2. Floodplain attributes. Edit model components and layer attributes using shape files. Create ARF, Streets and Levees.*

14:15 – 15:00 *Volume conservation, flood hydrology, and unconfined flooding. Rainfall and inflow hydrographs. Using Green Ampt and SCS-CN infiltration.*

15:00 – 15:15 ***Break***

15:15 – 15:45 *Hands-on session 3. Rainfall Lesson 7. Enter and edit rainfall data. (Demo)*

15:45 – 16:00 Review FLO-2D *.DAT and *.OUT files. Introduction to Urban modeling.

16:00 – 17:00 *Hands-on session 4: Lesson 11. Urbanized floodplain details. Set up streets and buildings.*

Friday, June 28, 2013
Channel Flood Routing, Mapping and Troubleshooting

8:00 – 8:30	*Channel flood routing overview. Channel/floodplain flow exchange.*
8:30 – 9:00	*Overview of GDS river channel tools. Introduction to Lessons 3, 5 and 8.*
9:00 – 9:45	*Hands-on session 5: Lessons 3, 4 and 5. Using GDS to create a simple rectangular channel. Interpolating the channel cross sections and slope and editing the bank elements in PROFILES.*
9:45 – 10:15	*Channel routing problems: What to look for? NOFLOCs, channel profiles, n-values, surging.*
10:15 – 10:30	***Break***
10:30 – 11:15	*Hands-on session 6: Lesson 8. Import a HEC-RAS project and create a natural channel in the GDS.*
11:15 – 12:00	*Using hydraulic structures in the urban environment: weirs, bridges and culverts for rivers and floodplains*
12:00 – 13:00	***Lunch***
13:00 – 13:30	*Mapper overview. Creating high resolution flood delineation maps.*
13:30 – 14:15	*Hands-on session 7: Mapper (Demo Session) Display flow depths, velocities, and water surface elevation maps. Create shaded contour maps and import aerial photos.*
14:15 – 14:30	*Troubleshooting overview, tools and methods for finding data errors.*
14:30 – 15:00	*Hands-on session 8: Troubleshoot a project.*
15:00 – 15:15	***Break***
15:15 - 15:45	*Optimization: Is the flood simulation running ok? Solving common problems: Volume conservation, numerical surging, sticky grid elements, and limiting Froude numbers.*
15:45 – 17:00	*Hands-on session 9: Create a project from scratch.*

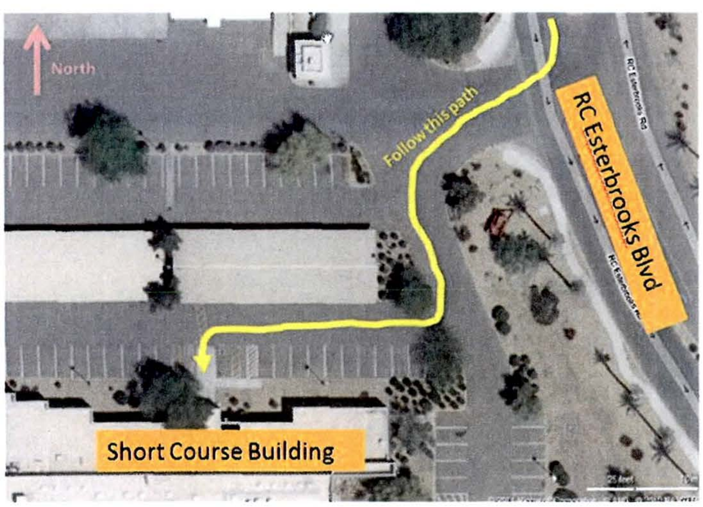


Table of Contents

INTRODUCTION

This document is organized as step-by-step instructions to create and run a detailed FLO-2D flood routing simulation. The lessons will guide the user through building a spatially variable model with infiltration, channel, levee, building and street components. The objective is to apply the Grid Developer System (GDS) to create a simple overland flow model that will be expanded with more channel and floodplain details. A comprehensive lesson on reviewing FLO-2D output data and creating flood maps using the MAPPER program completes the lessons.

Use this Workshop lesson book to compliment the FLO-2D, GDS and MAPPER manuals and other supporting documentation. There are review questions at the end of each individual lesson. Many of the questions reflect those technical support questions that FLO-2D users frequently pose. It is our intent to make the FLO-2D modeling system as user friendly as possible.

LESSON 1 – GDS GETTING STARTED

Overview

This lesson will shows the steps of generating a basic overland flood simulation with the FLO-2D model using the Grid Developer System (GDS) pre-processor program. Learn how to create the grid system, import and interpolate terrain elevations, assign inflow and outflow elements, import aerial photos as background and start a flood simulation.

Required Data

The lesson makes use of terrain elevation data, an inflow hydrograph, and aerial photograph provided in the installation folders.

File	Content	Location*
dtm.pts	Digital Terrain Model file	*Example Projects\Lesson 1*
c136.out	HEC-1 hydrograph	
Goat.tif	Aerial photographs	

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Check these folders to ensure the data is available before starting the lesson.

Step-by-Step Procedure

To run a FLO-2D flood simulation using these steps.

- Open the Grid Developer System (GDS) program;
- Import terrain elevation data;
- Create the Grid;
- Import aerial images;
- Outline the project area boundaries "computational domain";
- Interpolate the digital terrain elevation data and assign the grid element elevations;
- Assign hydrographs to selected inflow element;
- Select outflow grid elements;
- Run the FLO-2D model.

Step 1: Open the Grid Developer System program

Double click the GDS Basic icon from the desktop. 

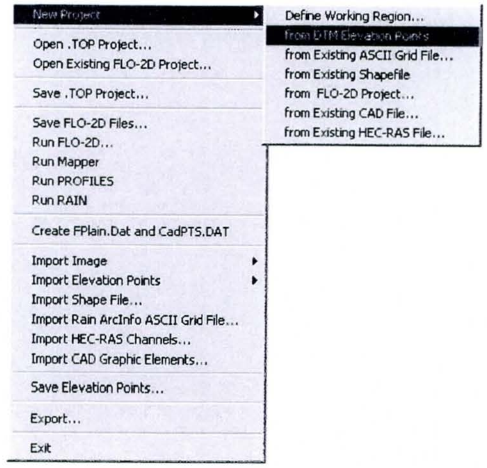
Step 2: Import terrain elevation data

The elevation data is typically in an text format where the elevation points are given as a table of space or comma separated x y z data.

6413334.59	1936112.79	81.81
6413388.62	1936112.00	81.57
6413500.78	1936108.47	81.29
6413664.70	1936102.41	80.95
6413784.88	1936094.97	80.75

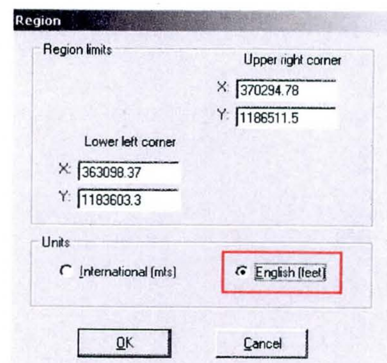
The first two columns are the x- and y- coordinates and the third column is elevation (ft or m). Other data formats are also supported (see the GDS Manual).

To import the file, click *New Project/from DTM Elevation Points* on the *File* menu.

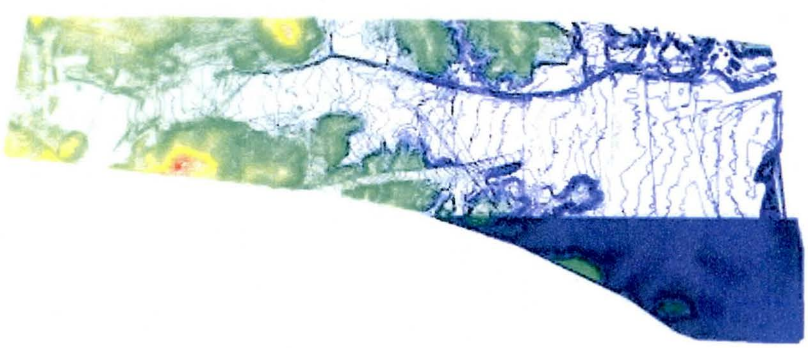


The GDS searches for a default file type with *.pts extension. Select the file *dtm.pts* and click *OK*. Choose the *English* system of units and click *OK*.

NOTE: A DTM points file must have a *.pts file extension and an ASCII grid data file must have a *.ASC extension.

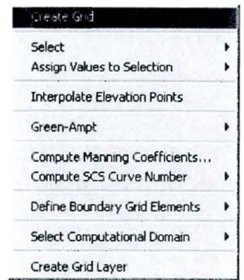


A dialog box of the number of points will appear. Click *OK* and the elevation data will be displayed with the view automatically zoomed to the full extent of the coordinates.



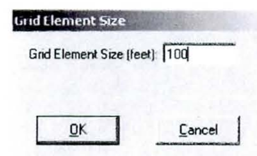
Step 3: Assign the grid element size and create the grid system

To create a new FLO-2D grid system, click the *Create Grid* command on the *Grid* menu.



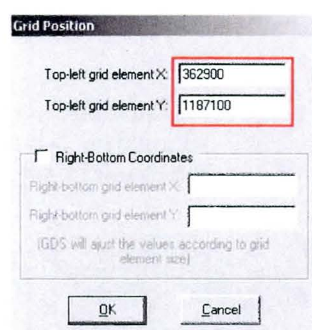
Input a grid element size (square grid) for the project and click *OK*.

> RECOMMENDATION: The peak discharge divided by the surface area of 1 grid element should be less than 3 cfs/sq. ft. or 1 cms/sq. m.
>
> $Q_{peak}/A_{surf} < 3$ cfs/sq. ft.



The grid element size should reflect the topographic data resolution and result in an acceptable number of total grid elements. A small grid element size enhances the FLO-2D model resolution but also increases the computer runtime and memory requirements. A balance should be sought between the grid element size (number of grid elements), resolution of the desired FLO-2D results and the computer runtime.

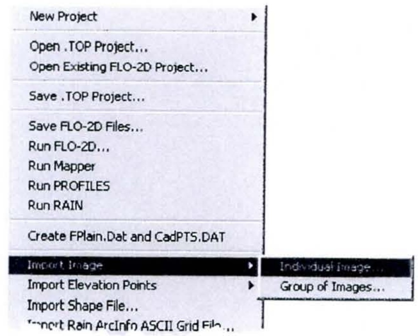
Assign a *Grid Position* and click *OK*.



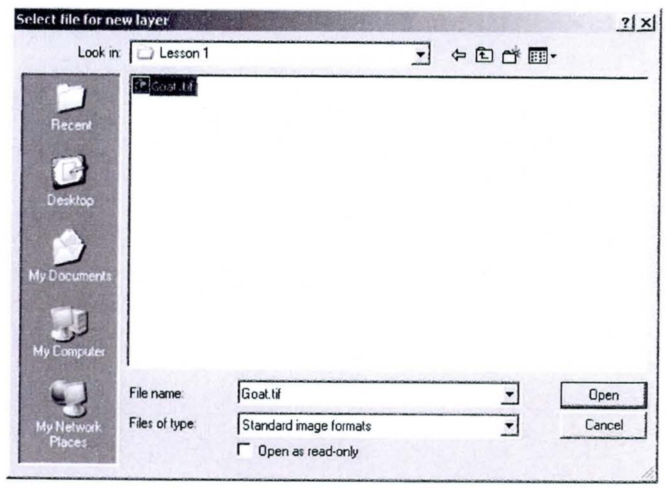
> SUGGESTION: Assign the upper left hand corner grid element coordinates as an even number (e.g. round to the nearest hundred).

Step 4: Importing aerial images

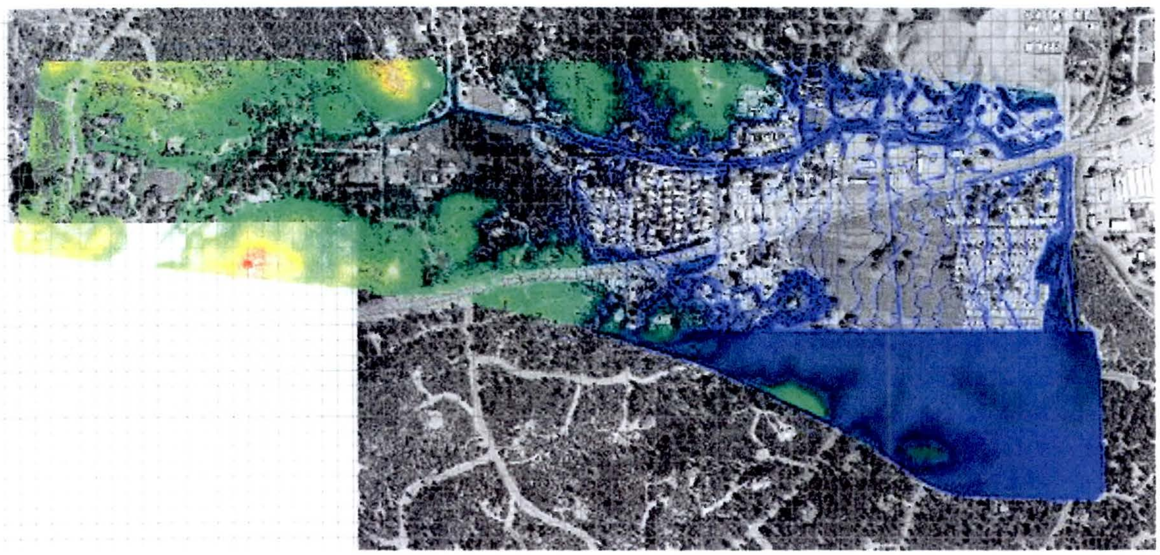
Importing background images will help to visualize the computational domain and project area with respect to the grid system. To import a background aerial photo use *Import Image/Individual Image…* in the *File* menu.



The image for this example project is located in the subdirectory *Example Projects\Lesson 1.* Images can be selected one by one or several at a time using the *Shift-click* or *Ctrl-click* on the files.

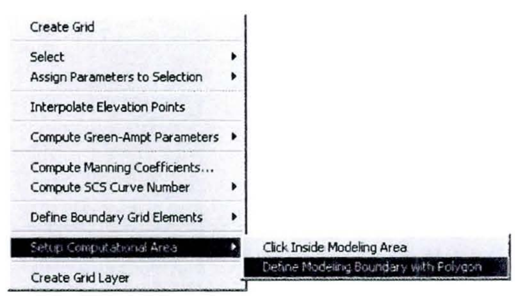


The following figure displays the Goat example project with background aerial photos and the elevations points.

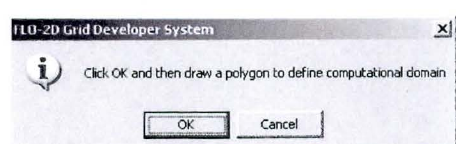


Step 5: Outline the project area boundaries

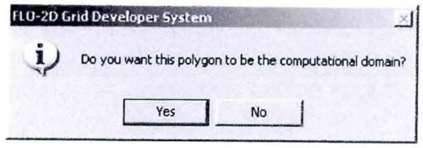
The computational domain can be defined by marking or highlighting the boundary grid elements. The grid boundary is defined by clicking *Define Modeling Boundary with Polygon* option of the *Set-up Computational Domain* in the *Grid* menu.



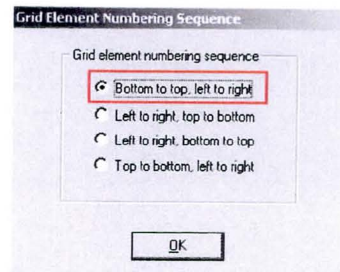
Click *OK* on the dialog box that appears instructing the user to draw the boundary polygon.



Draw a polygon by clicking on each polygon vertex along the desired boundary. The polygon is completed by double clicking on the last vertex. The system displays the message.

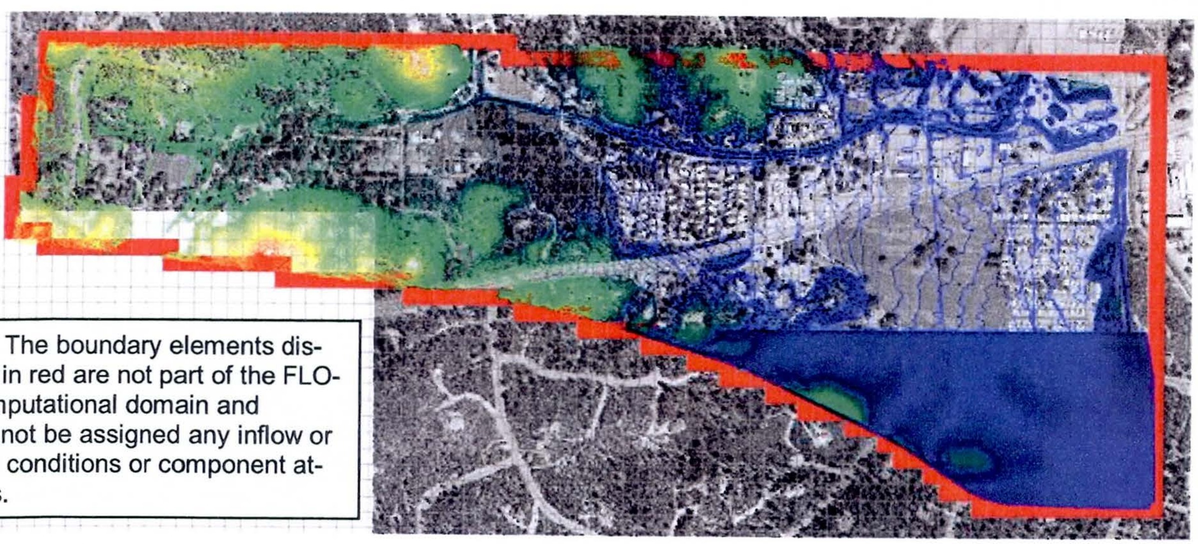


A *No* response will erase the polygon while a *Yes* response will proceed with creating the computational domain. A dialog box will appear so that the grid system numbering scheme can be selected.



The numbering sequence can help in locating grid elements later. It is suggested to have the number sequence increase in the flow direction. A recommended numbering scheme is left to right, top to bottom if the drainage system runs from north to south or from top to bottom of the screen.

Click *OK* to generate the computational domain grid system.



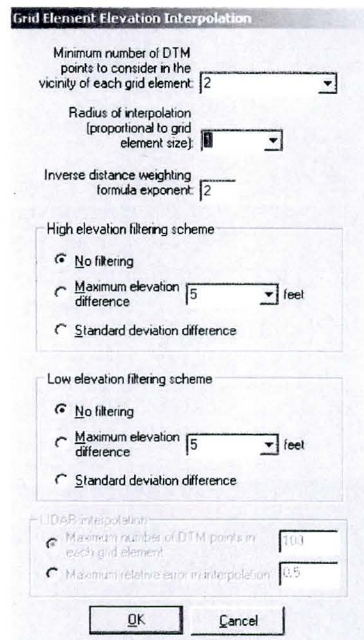
NOTE: The boundary elements displayed in red are not part of the FLO-2D computational domain and should not be assigned any inflow or outflow conditions or component attributes.

The FLO-2D grid system now consists of all the grid elements within the computational domain.

Step 6: Interpolate the digital terrain elevation data & assign grid element elevations

A set of DTM points may be randomly distributed in the flow domain and some grid elements may have many DTM points or none at all. The FLO-2D model requires that each grid element be assigned a representative elevation.

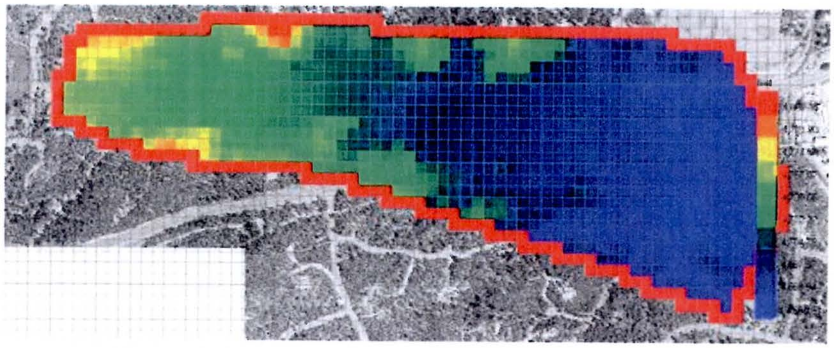
To interpolate and assign the grid element elevations from the random DTM points or contour point elevations, click the *Interpolate Elevation Points* command on the *Grid* menu. A dialog box will display the elevation interpolation parameters.



The user can adjust the interpolation criteria using the *Grid Element Elevation Interpolation* dialog box. Click *OK* to interpolate the DTM points using the default interpolation criteria.

Using *View/Grid Element Elevation Rendering,* the interpolated grid element elevations are shown by color. Red/orange represents the highest elevation while blue the lowest elevation in the computational domain.

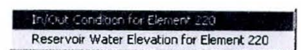
Turn off the Elevation Rendering by *clicking View/Grid Element Elevation Rendering*. Turn off the DTM points by clicking *View/Elevation Points*.



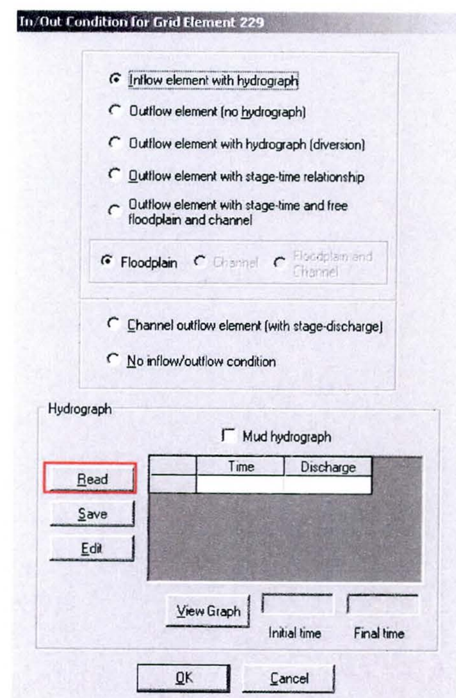
Step 7: Assign hydrographs to selected inflow nodes

Use this command to define inflow and outflow grid elements. Right click on the grid element that was chosen to be an inflow node, and select the *In/Out Condition for Element* **** from the sub-menu.

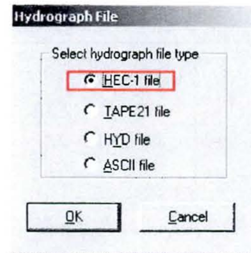
NOTE: Your element may not be #220.



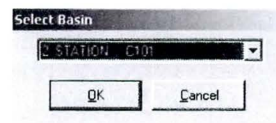
The flowing dialog box allows editing the inflow conditions.



Click on both the *Inflow element with hydrograph* and the *Floodplain* radio buttons to assign an inflow hydrograph to the selected grid element. Then click *Read* to display the following dialog box to import a hydrograph with HEC-1, Tape21, HYD or Text file formats.

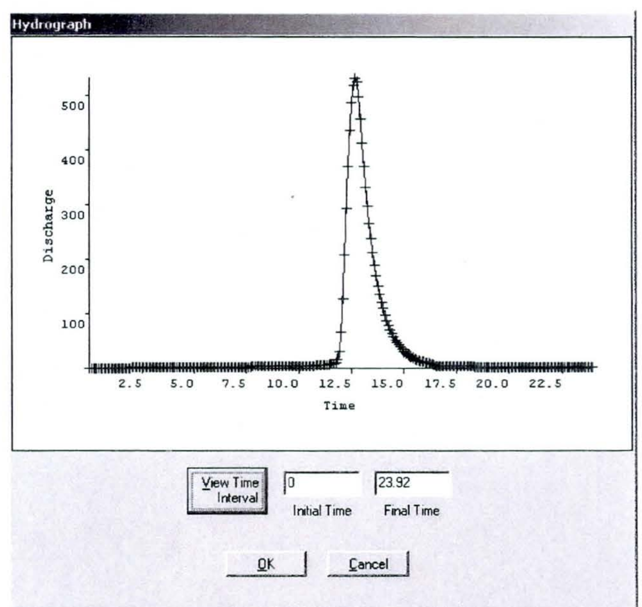


Click the *HEC-1 file* option and search for the *c136.OUT* file in the *ExampleProjects\Lesson 1* folder. Then select *2 STATION C101* hydrograph from the dropdown list.



After selecting the HEC-1 hydrograph and clicking *OK*, the hydrograph data is loaded into the data table in the dialog box. Click *Save* to create a test.hyd file. These files are text files generated by the GDS to save the hydrograph data and allow later recovery of hydrographs when a project is read from a *.TOP file. The .hyd file is not used by the FLO-2D model.

Now click *View Graph* to plot the hydrograph in the following window.

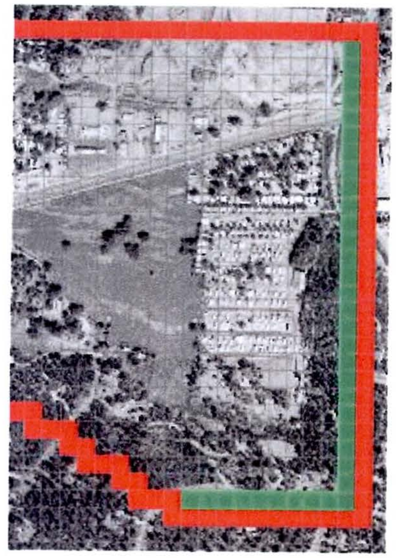


Inflow grid elements with assigned hydrographs are displayed in a cross-hatched green pattern in the GDS for identification.

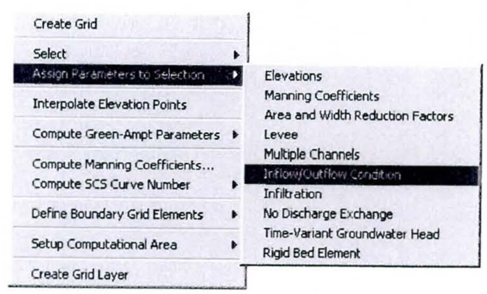
Step 8: Select outflow grid elements

The FLO-2D grid system represents an impermeable border from which no flow will escape. The flow will pond against the boundary unless outflow nodes are assigned. Outflow elements discharge any flow off the grid system without effecting the water surface elevation. The outflow elements approximate normal depth flow conditions from upstream elements. To assign the outflow elements, first click the *Select Element by Element* icon in the *GDS* tool bar, then click on each element (or hold the shift key while dragging the mouse to select multiple elements).

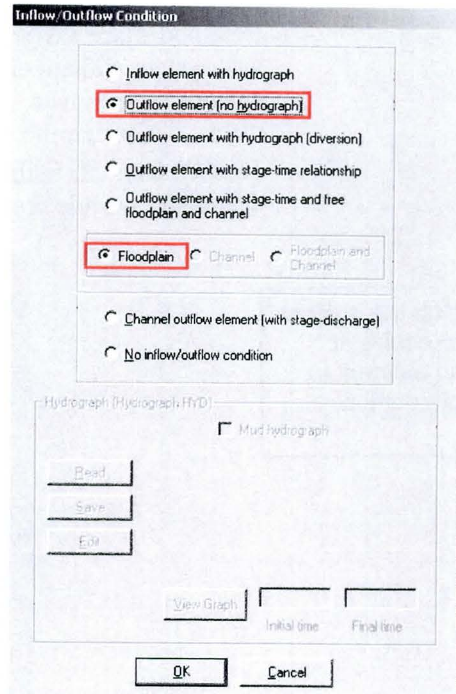
NOTE: Avoid doubling up the outflow nodes. Outflow nodes require at least one upstream grid element to compute a normal depth outflow condition.



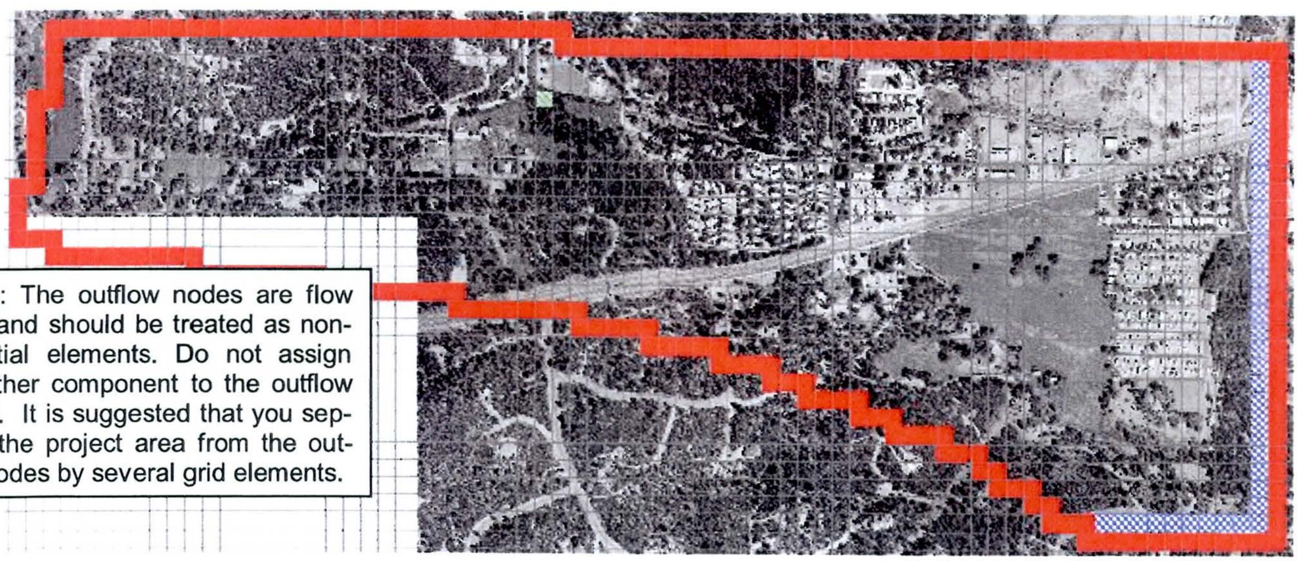
Now click the *Assign Parameters to Selection/Inflow/Outflow Condition* command in the *Grid* menu



Select *Outflow element/(no hydrograph)* and *Floodplain* radio buttons and click *OK*.



The outflow elements are identified with a blue cross-hatched pattern.

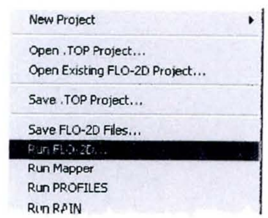


NOTE: The outflow nodes are flow sinks and should be treated as non-essential elements. Do not assign any other component to the outflow nodes. It is suggested that you separate the project area from the outflow nodes by several grid elements.

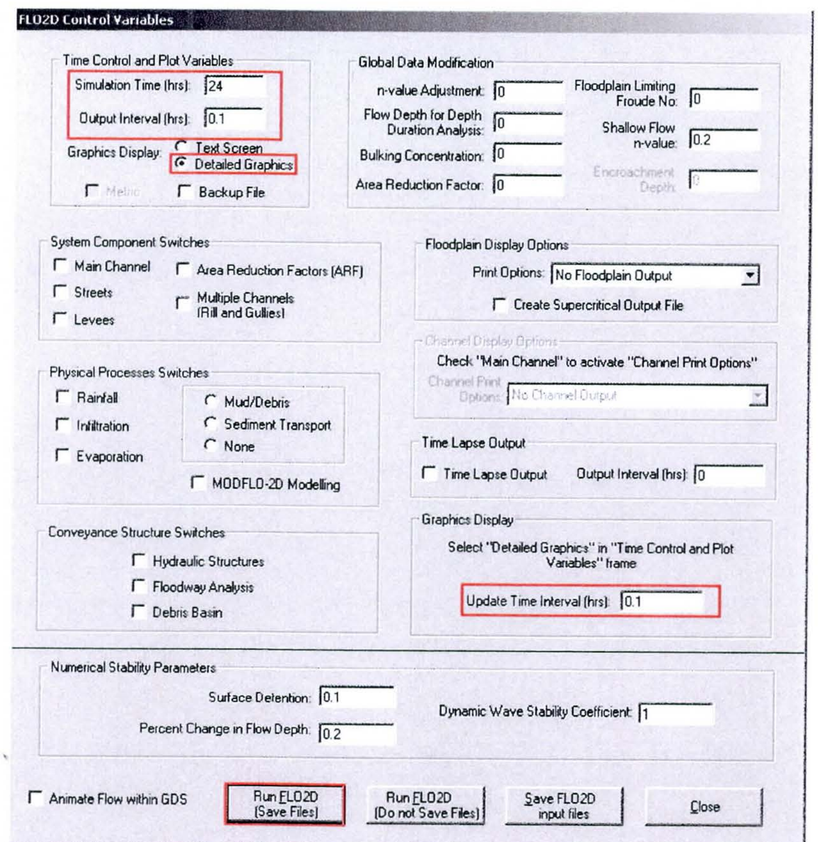
Step 9: Save project and run the FLO-2D flood simulation.

Create a new folder in the Example Projects folder named GDS Test and save the FLO-2D data files to it. Avoid overwriting any of the Example Projects files. To Save the FLO-2D flood simulation, use the *Run FLO-2D...* model command in the *File* menu.

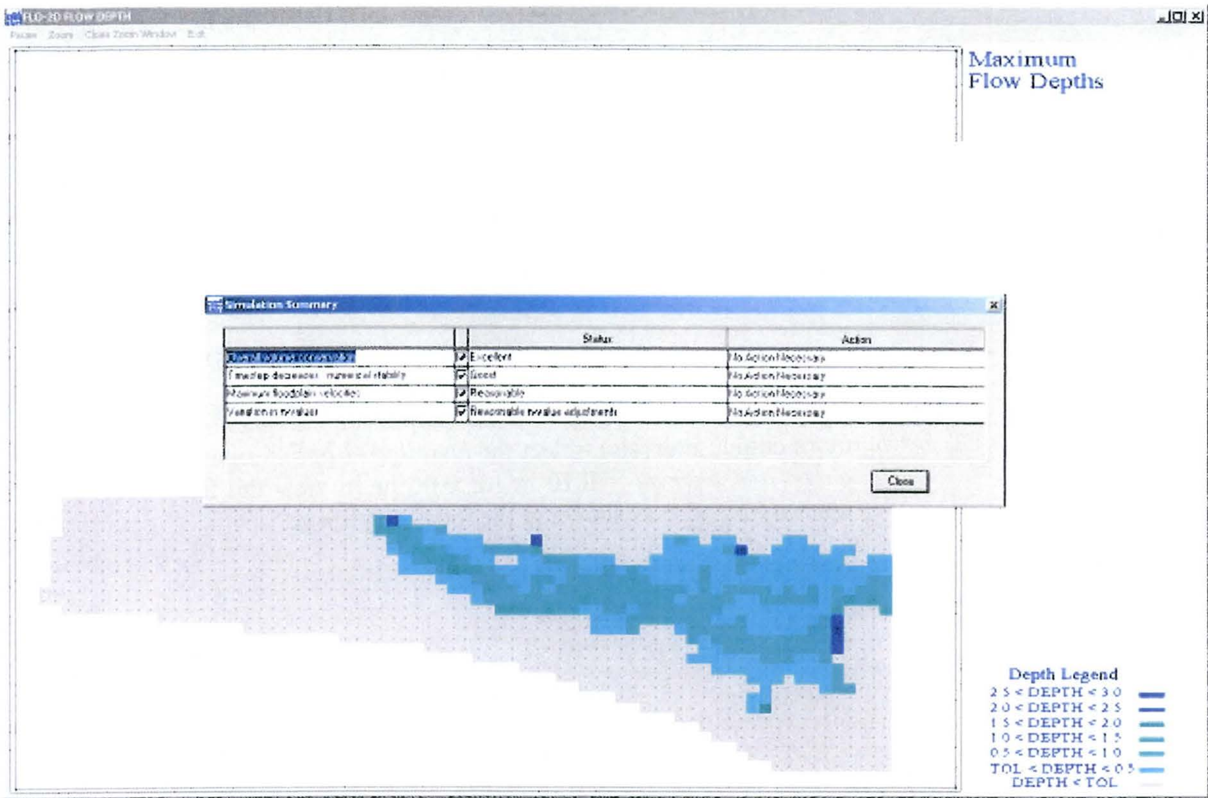
NOTE: You may get a message indicating that the outflow nodes have elevation inconsistencies. The model will automatically fix this for you during the preliminary data file check.



Input the *Simulation Time* = 24hr, *Output Interval* 0.25 hr (or 0.10 hrs for more output intervals), select the *Detailed Graphics* option and input *Update Time Interval* = 0.10 hr (or 0.05 hr to view the flood progression more frequently) as shown in the following image.



Now click *Run FLO2D*. Monitor the floodwave progression, the inflow hydrograph plot, simulation time, volume conservation, and area of inundation.



Summary

This is the completion of a basic overland flood simulation using the FLO-2D model along with digital terrain elevation data and an inflow hydrograph. This lesson has demonstrated how to create the grid system, import the DTM data, interpolate the grid element elevations, assign inflow and outflow elements, and import background aerial images.

To add more detail and components to the project such as channels, streets, levees and buildings, complete the following lessons.

Review Questions

1. What are the five methods to start a FLO-2D project in the GDS?
   i. ______________________________
   ii. ______________________________
   iii. ______________________________
   iv. ______________________________
   v. ______________________________

2. What is the typical DTM data format and what type of file must be used?

   ____

3. What factors should be considered to balance simulation speed with modeling accuracy?

   ____

   ____

4. What is the criterion for selection the grid element size? ______________________________

5. What is the floodplain default n-value? Is this reasonable for most projects?

   ____

6. How is the grid element elevation determined from DTM data? ______________________________

   ____

7. Is the grid system boundary included in the FLO-2D computation domain?

   ____

8. Which of the following can be share a node with outflow conditions? Check box.

   ☐ Streets ☐ Levees

   ☐ Buildings ☐ Bridges/Culverts

   ☐ Channels

9. True or False Double-up the outflow nodes the model will calculate twice the outflow discharge?

10. What is the difference between the Outflow Interval and the Update Time Interval on the Control Panel?

    ____

    ____

LESSON 2 – GDS COMPONENT EDITING

Part 1. Floodplain Attributes

Topography and Roughness

Overview

This lesson will use the GDS to graphically edit FLO-2D grid element attributes such as n-values and elevations. It will also show how to compute n-values attributes by interpolation from shape file.

Required Data

The lesson requires the use of FLO-2D data files, n-value shape files and aerial photographs provided in the installation folders as follows.

File	Content	Location*
**.DAT*	FLO-2D Data Files	*Example Projects\Lesson 2_1*
Goat.tif	Aerial photographs	*Example Projects\Lesson 1*
n value.shp	N-value shape file	*Example Projects\Lesson 2_1*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

Graphically edit floodplain attributes for the FLO-2D model using the GDS by following these steps.

- Open the Grid Developer System (GDS) program;
- Load the *Lesson 2_1* project;
- Import aerial images;
- Select and deselect grid elements;
- Graphically edit floodplain elevation;
- Import n-value shape file;

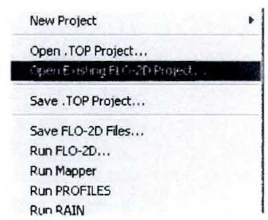
- Graphically edit n-values;
- Compute and view Manning's coefficients;
- Save the FLO-2D data files.

Step 1: Open the Grid Developer System program

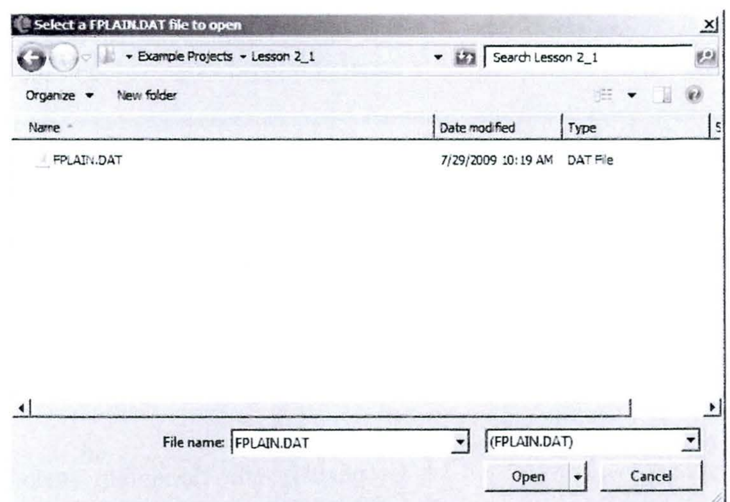
Double click the GDS Basic icon from the desktop. 

Step 2: Load the model Lesson 2_1

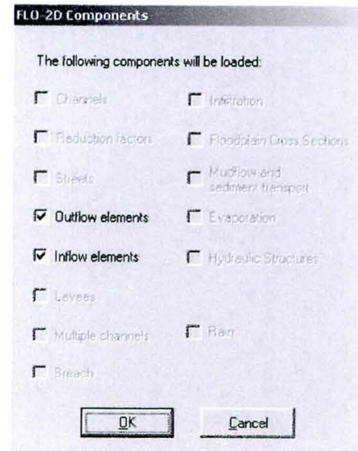
Click the *File* menu in the GDS and select the *Open Existing FLO-2D Project* command.



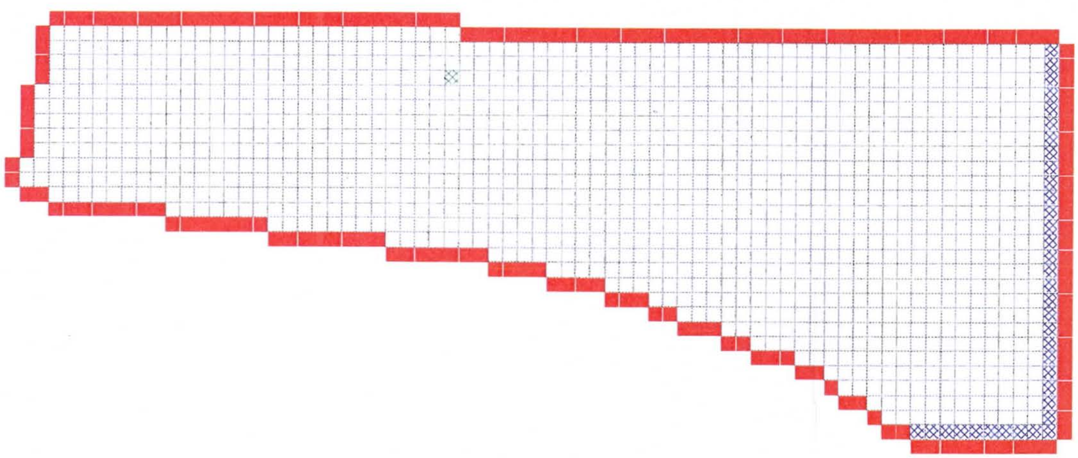
Browse the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation\Example Projects\Lesson 2_1* and double click the file *FPLAIN.DAT*.



The following dialog box will be displayed. Click *OK* to load the model and display the grid system.



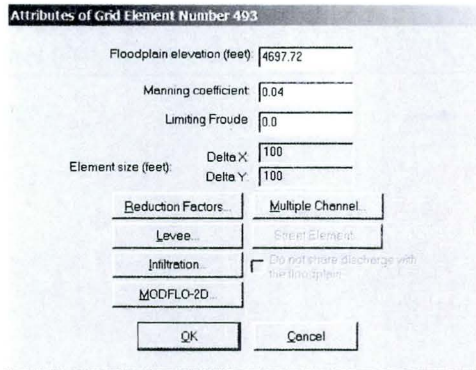
The display of the grid system should look like the following image.



Step 3: Importing aerial images (see Lesson 1 - Step 4)

Step 4: Select and deselect grid elements for editing

There are several ways to select grid elements for component editing. To select a single element, *double click* it and load the following dialog box. Click *Cancel* to close the box.



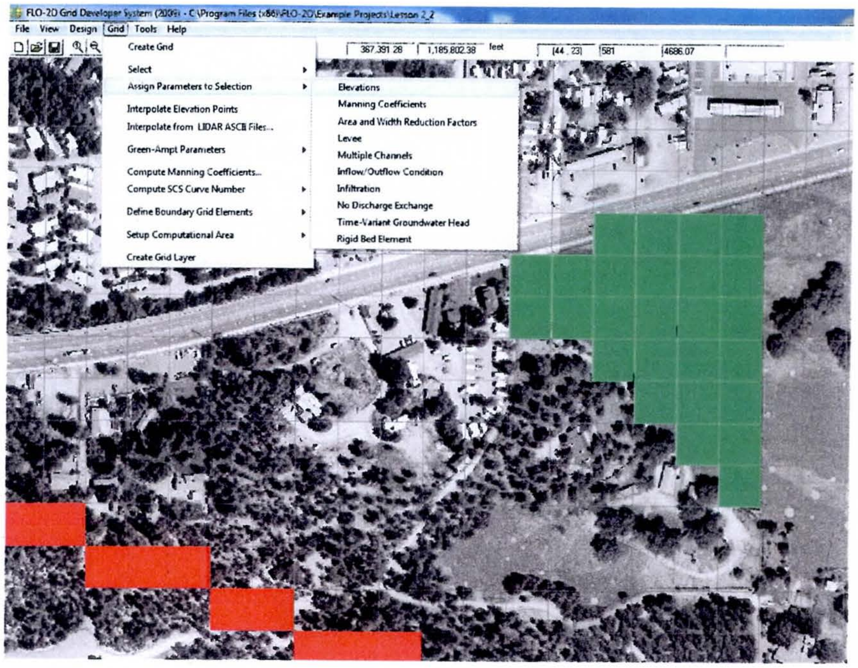
Click the *Select Element by Element* button on the toolbar. Click on several grid elements and notice selected elements have green cross hatches. Select multiple grid elements by holding down the *shift* key while dragging the mouse over the grid elements. Deselect single elements by clicking the element again. Deselect multiple elements by holding down the *Ctrl* key while dragging the mouse over undesired elements.

Select an area of grid elements by clicking the *Select Element Defined by Polygon* button on the tool bar. Click the *Unselect Element* button on the toolbar to deselect all grid elements.

Step 5: Graphically edit floodplain elevations

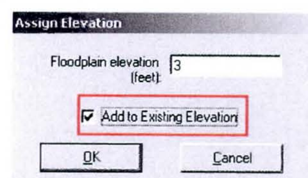
To edit the elevation of a single grid element, just double click the element. Change the elevation parameter in the dialog box and click OK.

Zoom in on the area of bare ground south of the highway. To edit the elevation of several grid elements, click the Select Element by Element button and select several grid elements. Click the Grid/Assign Value to Selection/Elevations.



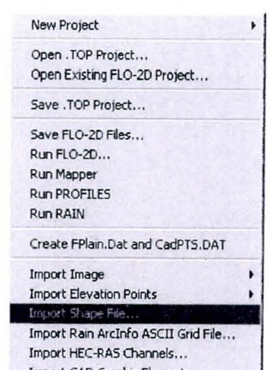
Enter a *3* in the dialog box, check the *Add to Existing Elevation* check box and click *OK*.

Note: Assign a single elevation to a group of grid element. This will facilitate the design of a graded surface such as a parking lot.

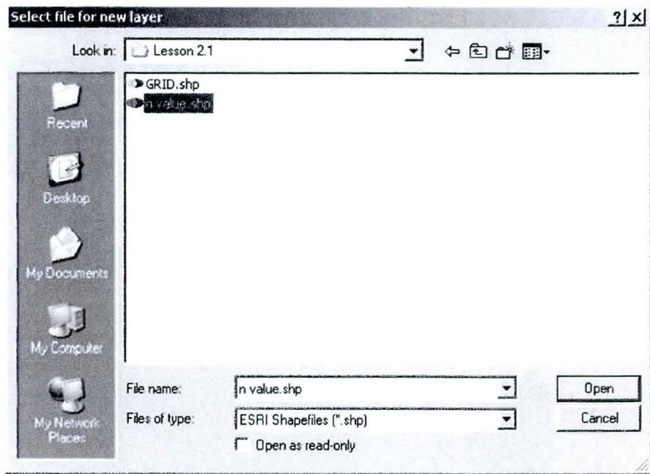


Step 6: Import n-value shape file

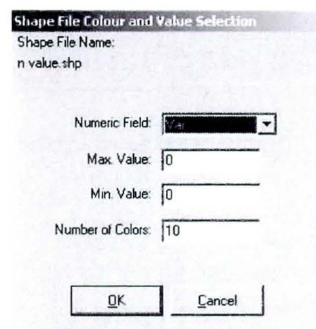
From the *file* menu, click *Import Shape File…*



In the *Lesson 2.1* folder, open the *n value.shp* file.

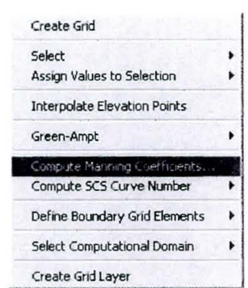


Click *OK* on the following dialog box to load the shape file.

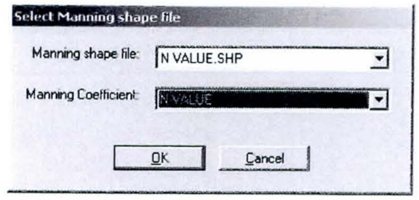


Step 7: Compute and view Manning's coefficients

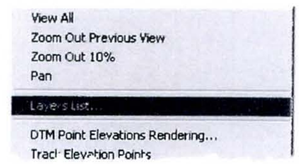
From the *Grid* menu, click *Compute Manning's Coefficients…*



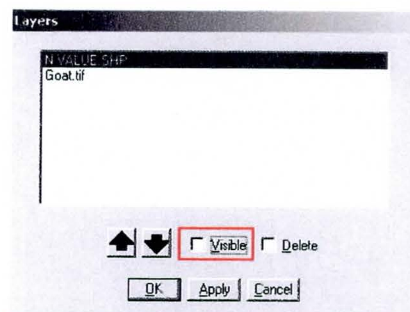
Change the *Manning Coefficient* from *VAR* to *N-VALUE* and click *OK*.



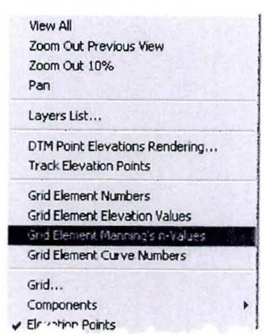
Turn off the shape file by clicking *View/Layers List*.



Select the *N Value.SHP* and click the *Visible* checkbox. Click *OK* or *Apply* to hide the shape file. The arrow buttons are used to specify the top layer.



To see the n-value changes, click *Grid Element Manning's n-Values* from the *View* menu.

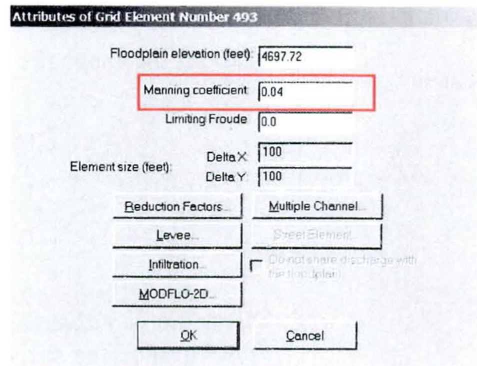


Zoom in on a portion of the highway and pan around the grid to view the different n-values. Repeat the procedure to turn off the grid element n values.

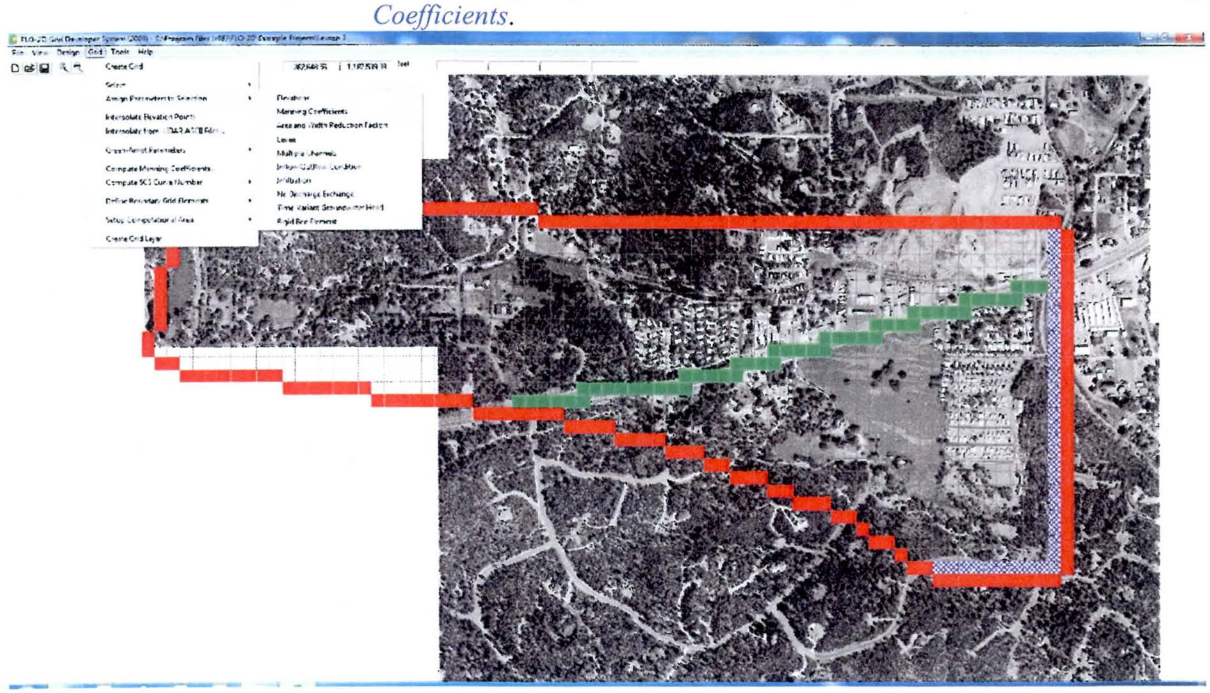
Step 8: Graphically edit n-values

To represent a street without curbs, the floodplain n-values can be adjusted. Double click a grid element on the highway shown in the aerial photograph. Adjust the Manning's n-value to reflect the highway roughness.

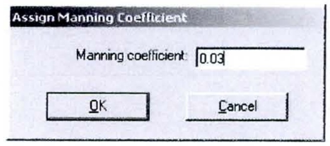
Note: Other floodplain attributes can be accessed from this dialog box. The grid element elevation can also be edited.



Use the *Select Element by Element* Button to select the grid elements along the highway. Then click *Grid/Assign Values to Selection/Manning Coefficients*.



Enter an n-value that represents the roughness of the highway into the *Assign Manning Coefficient* dialog box and click *OK*.



Step 9: Save project

To save the project click the *Save FLO-2D Files…* command in the *File* menu.

Create a new folder in the Example Projects folder named *Test 1* (or any folder name) and save the data files to it. Avoid overwriting any of the Example Projects files. If a file is overwritten inadvertently, the files can be recovered from the installation files.

Review Questions

1. Name three methods to revise grid element elevations?

____

____

____

2. Overland flow details are attributes that are assigned to all grid elements spatially. Which of the following are overland flow attributes? Check all that apply.

- ☐ Elevation
- ☐ Floodplain cross sections
- ☐ N-value
- ☐ Infiltration

3. Name three methods to select and deselect grid elements?

____

____

____

4. Identify the following buttons?

 ______________________

 ______________________

 ______________________

 ______________________

 ______________________

 ______________________

 ______________________

 ______________________

5. The following technical support files are found in the FLO-2D folder under c:\Program Files\FLO-2D? Check all that apply.

- ☐ Example Projects
- ☐ User's Manuals
- ☐ Tutorials
- ☐ Pocket Guide
- ☐ PowerPoint Presentations
- ☐ License Documentation

6. Which data file is the grid element elevation and n-value attributes written to?

- ☐ CONT.DAT
- ☐ TOLER.DAT
- ☐ CADPTS.DAT
- ☐ INFIL.DAT
- ☐ FPLAIN.DAT
- ☐ CHAN.DAT

7. How can a FLO-2D data file (*DAT) be viewed?

____

____

Part 2. Urban Floodplain Attributes

Buildings, Streets, Levees and Infiltration

Overview

In Part 2 of this lesson shows how to add urban features to the model. These include buildings represented by area reduction factors (ARF) and width reduction factors (WRF) attributes, streets, levees and infiltration parameters. Save the FLO-2D data files after each editing step to avoid the significant loss of data.

Required Data

The lesson requires the use of FLO-2D data files and aerial photographs provided in the installation folders as follows.

File	Content	Location*
**.DAT*	FLO-2D Data Files	*Example Projects\Lesson 2_2*
Goat.tif	Aerial photographs	*Example Projects\Lesson 1*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

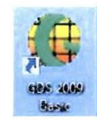
Step-by-Step Procedure

To graphically edit FLO-2D urban attributes using the GDS follow these steps.

- Open the Grid Developer System (GDS) program;
- Load the Lesson 2_2 project;
- Import aerial images;
- Graphically edit area reduction factors (ARF) and width reduction factors (WRF);

- Add a street segment;
- Create a levee;
- Graphically edit infiltration parameters;
- Save the FLO-2D data files and run the model.

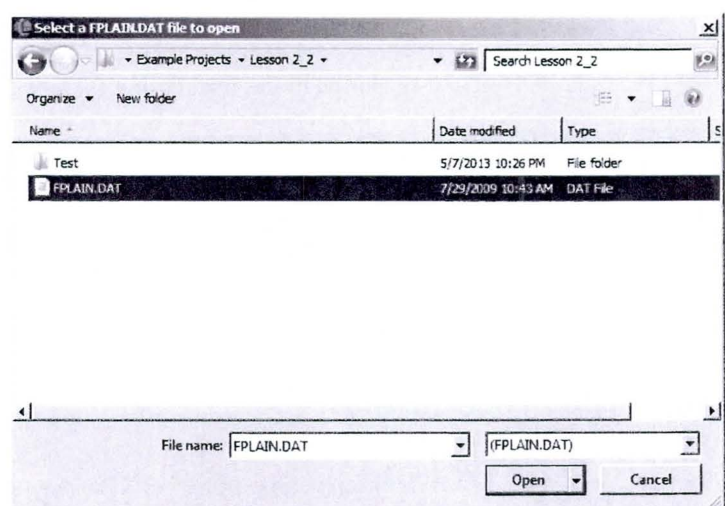
Step 1: Open the Grid Developer System program

Double click the GDS Basic icon from the desktop. 

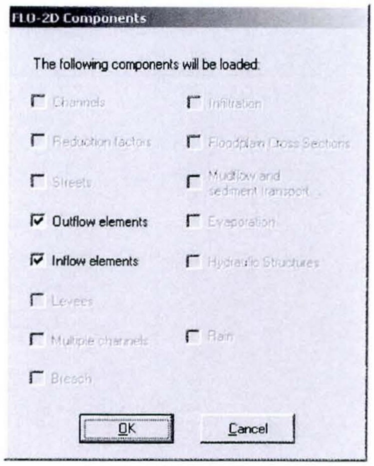
Step 2: Load the model Lesson 2.2

Click the *File* menu in the GDS and select the *Open Existing FLO-2D Project* command.

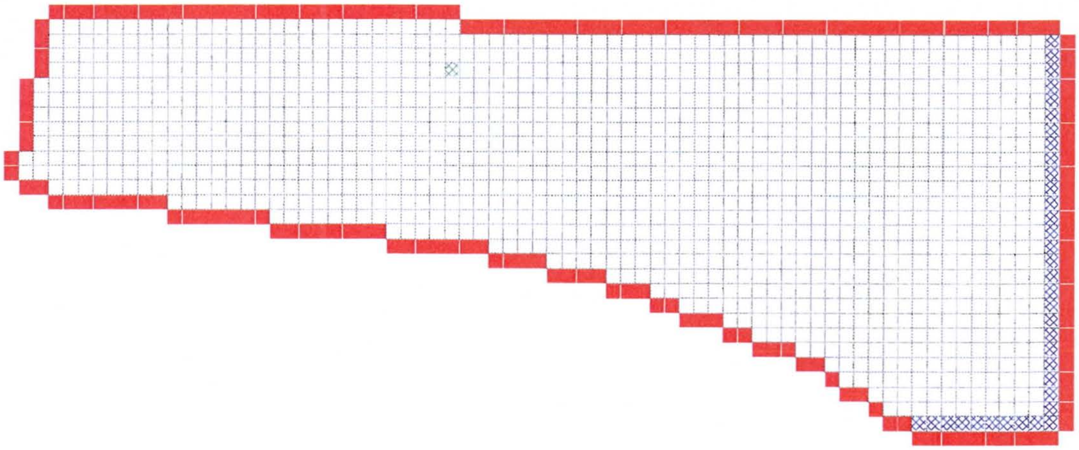
Browse to the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation\Example Projects\Lesson 2_2* and double click the file *FPLAIN.DAT*.



The following dialog box will be displayed. Click *OK* to load the model and display the grid system.



The display of the grid system should look like the following image.

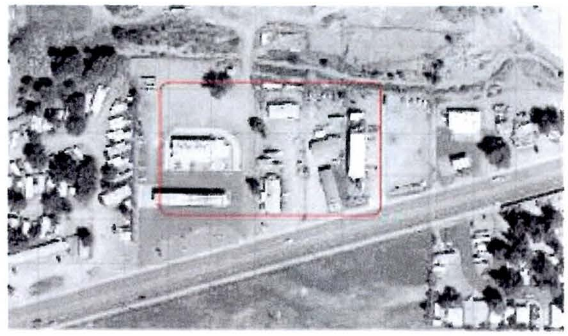


Step 3: Importing aerial images (see Lesson 1 - Step 4)

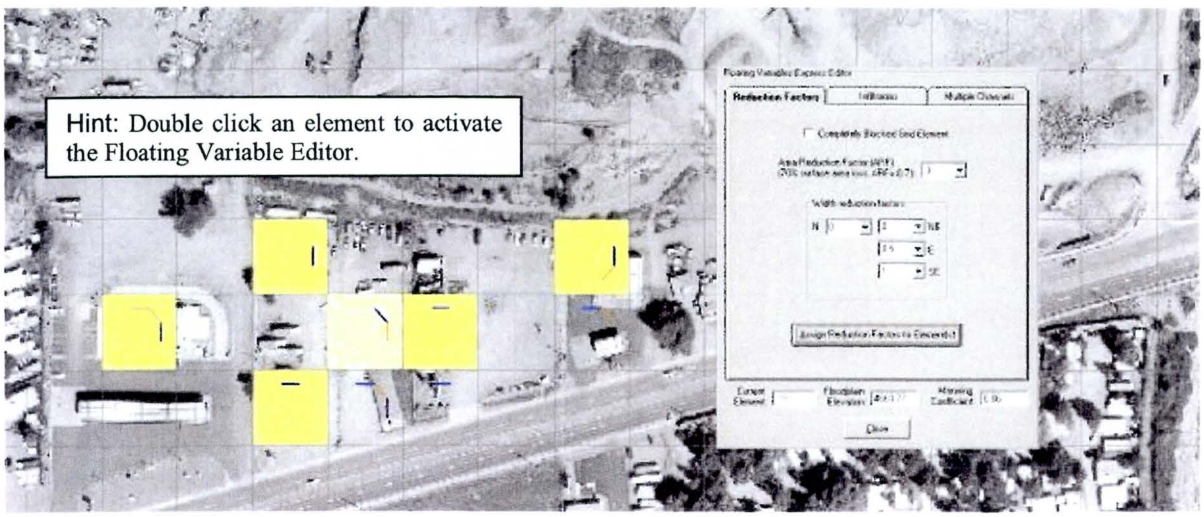
Step 4: Graphically edit Area Reduction Factor (AFR) and Width Reduction Factor (WRF) values

Single building assignment…

This assignment will simulate the loss of floodplain storage or flow obstruction due to buildings or other floodplain features. Zoom in on the populated area north of the highway and see several large rectangles. These features are buildings and a gas station. The thin rectangle at the bottom of the red box on the following image is a pump station and does not impact flood storage or flow. The buildings partially cover grid elements and will block the flow. Represent these features by editing ARF and WRF values of the coinciding elements.



Click *Tools/Floating Variable Editor* and click the *Reduction Factors* tab. Use this variable editor to assign reduction factors for the buildings.



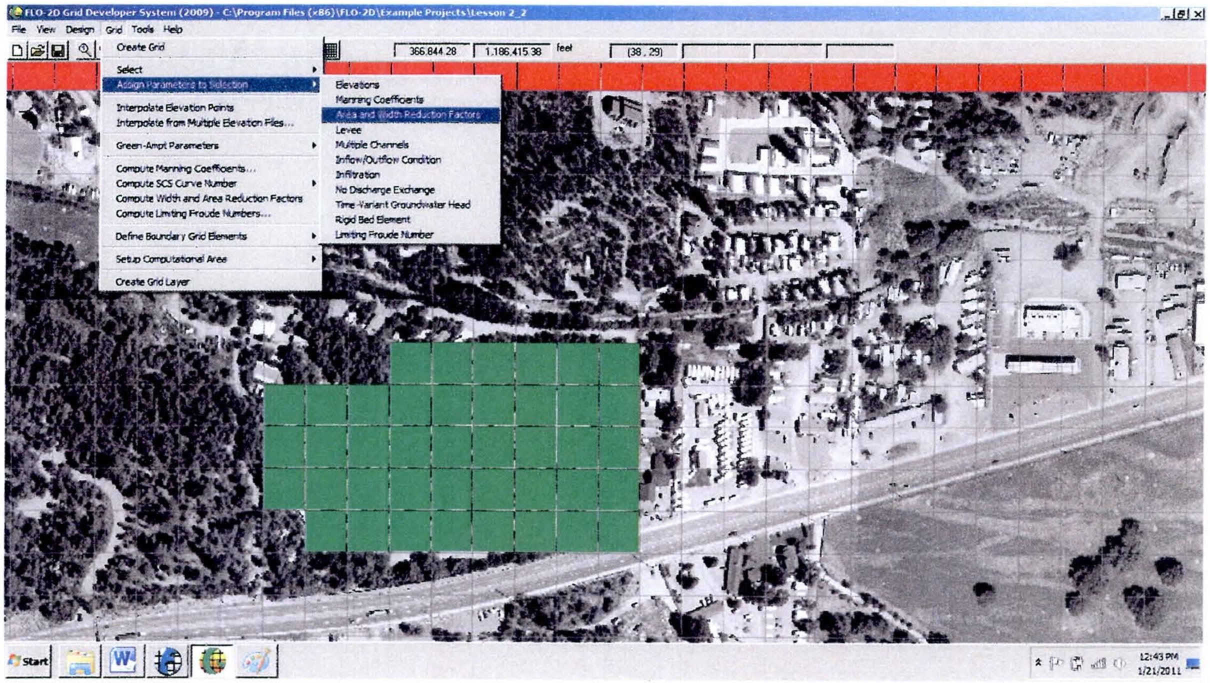
In this dialog box, it's possible to completely block the grid element (cell) from receiving any flow by checking the *Completely Blocked Cell* box. To simulate a partial loss of storage on the grid element due to buildings or other features, assign an *Area Reduction Factor (ARF)* from 5% (0.05) to 95% (0.95). If an ARF value less than 5% is assigned, the FLO-2D model will automatically set the ARF = 0.0. Similarly, if an ARF value greater than 95%, the FLO-2D model will automatically set the ARF = 1.0.

Note: The model automatically recognizes any WRF values from the opposite flow direction. For example, assign a WRF value for the east direction and it will be also be assigned as the west direction WRF of the grid element to the east.

The *Width reduction factor (WRF)* can be assigned in four directions (north, northeast, east and southeast). The other four directions are automatically assigned by the FLO-2D model from the contiguous grid element. The WRF values also vary from 0.0 to 1.0 (100% obstruction to flow). When assigning the WRF values, remember that the grid element is represented as an octagon for the eight flow directions.

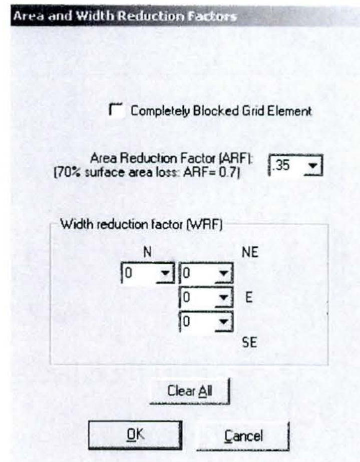
MULTIPLE building assignment…

Zoom in on the neighborhood north of the highway. Use the *Select Element Defined by Polygon* button to select the grid elements that coincide with the neighborhood.



Click *Grid/Assign Value to Selection/Area and Width Reduction factors* to load the following dialog box. Enter a *.35* into the box and click *OK*.

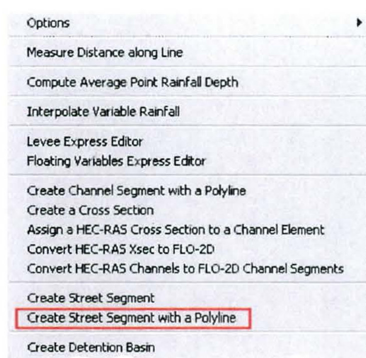
> Note: Do not assign WRF values using the multiple grid element option.



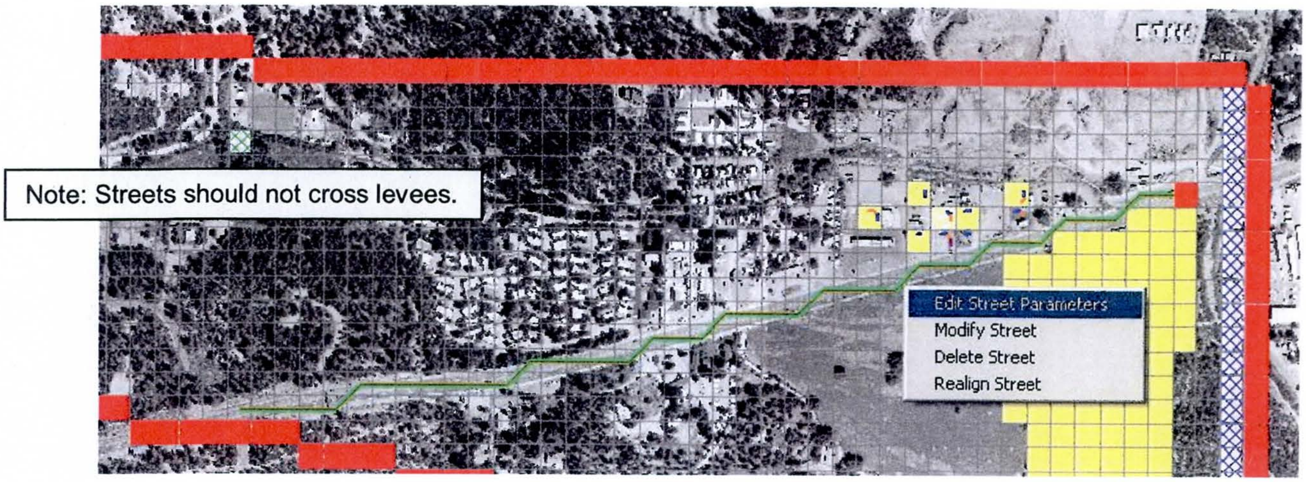
Save the FLO-2D files.

Step 5: Add a street segment

In this case the highway will be assumed to have a curb and gutter and will function as a flood conveyance feature. Zoom in on the highway and note that this is a two lane high-way but the 100 ft grid element will encompass both lanes and thus we will consider it to be a single street with the assumption that it has a width of 40 ft. Click on the *Create Street Segment with Polyline* option on the *Tools* menu.



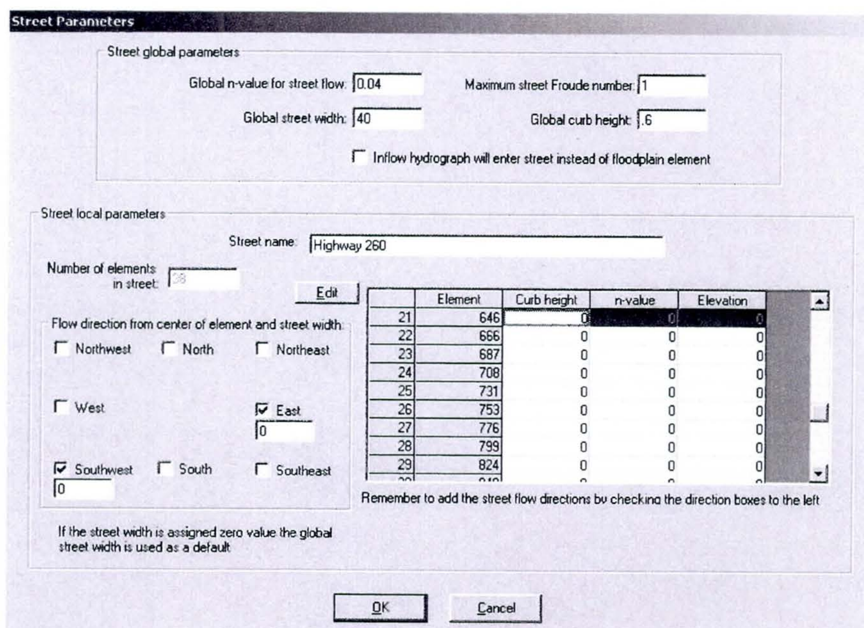
The cursor changes from an arrow to a cross. Click grid element where the highway starts on the left side of the grid system and drag the cursor along the street. Follow the street as closely as possible. In this case try and avoid street elements that meet at right angles. Double click the last street element to finish the street.



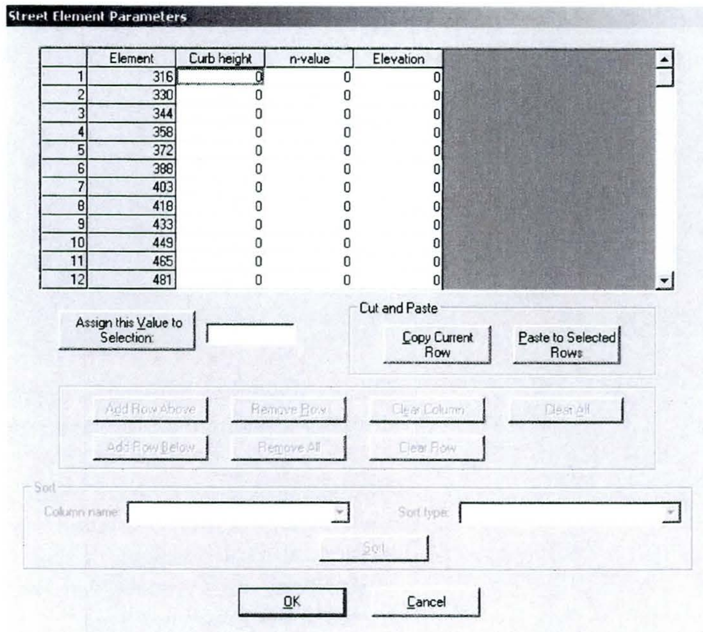
Once the street segment is created, define the street parameters by left clicking anywhere along the street. Click the *Edit Street Parameters*.

Edit the global parameters in the top area of the dialog box as shown below. Assign a *global n-value = 0.02*, street width, limiting street Froude number and curb height. These values will be assigned to all the streets unless they area superseded by individual grid element street assignments in the lower box area.

Assign a street name *Street name*. Street intersections can be modeled adding by additional directions within the grid element.

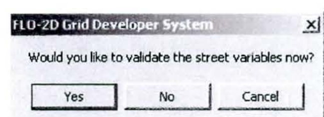


Click the *Edit* button to assign individual grid element street attributes (curb height, n-value and elevation) that will supersede the global assignment parameters.



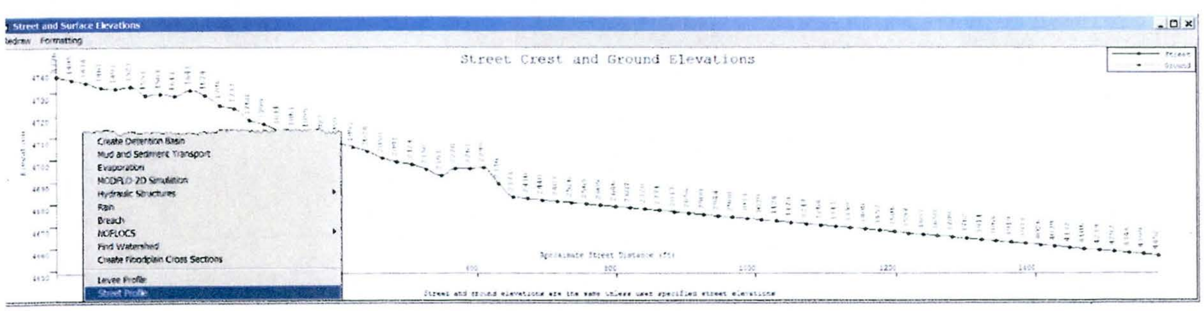
Note: Validating the table does not save the STREET.DAT file. It only saves the table in the GDS. You must still save the FLO-2D Files to create the STREET.DAT file.

Click *OK* to exit the *Street Element Parameters* and click *OK* again to exit the *Street Parameters* dialog box. Click *Yes* to validate the table for the GDS.



Save FLO-2D files.

To view a profile of the street, click *Tools/Street Profile*. Edit the elevation of grid elements that cause severe dips or mounds in the street profile unless the dip or mound accurately reflects the situation.



Step 6: Create a levee

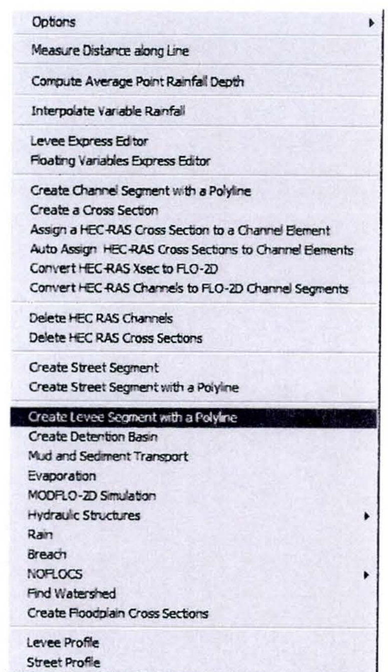
Levees can represent berms, roadways or railroad embankments, or floodwalls. Any linear feature that can obstruct the flow and yet still be overtopped can be modeled using a levee function.

Click *View/Components/Reduction Factors* to turn off the ARF's.

Zoom in on the neighborhood to the southeast of the highway and see that a brick wall is built between the homes and the highway. The task is to create a levee to represent this floodwall.

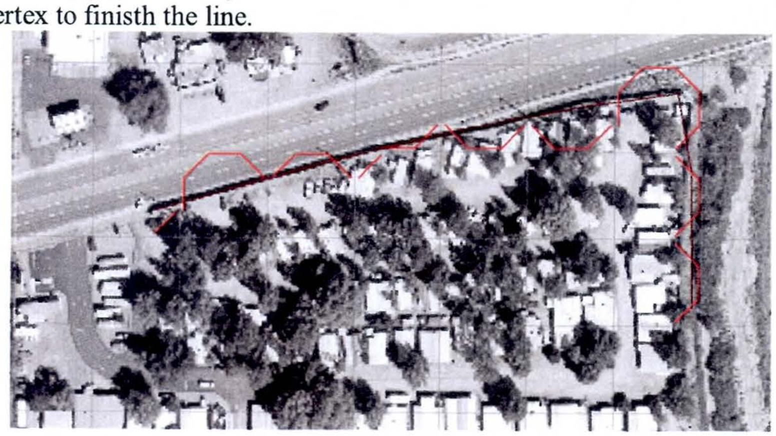
Click *Tools/Create Levee Segment with a Polyline*.

Note: Levees are assumed to have negligible width. If the storage loss due to levees was considered to be important, use ARF values in the levee node.



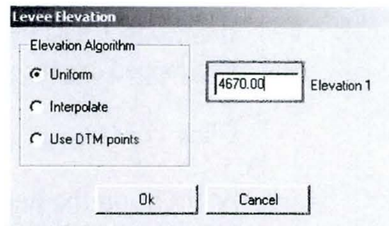
Draw a polyline along the wall to create a levee. Double Click the last vertex to finisth the line.

Warning: Do not put levees perpendicular to street elements.



Warning: Do not create redundant levee elements. For example, do not place a levee in the north direction of one grid element and then in the south direction of the grid element above it.

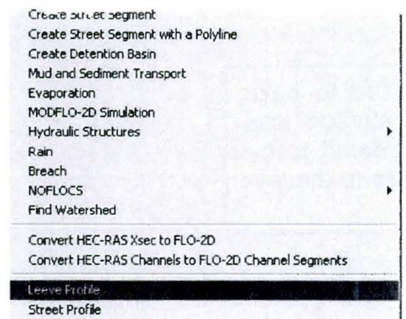
Enter a value of *4670.00* ft to set a *Uniform* crest elevation and click *OK*.



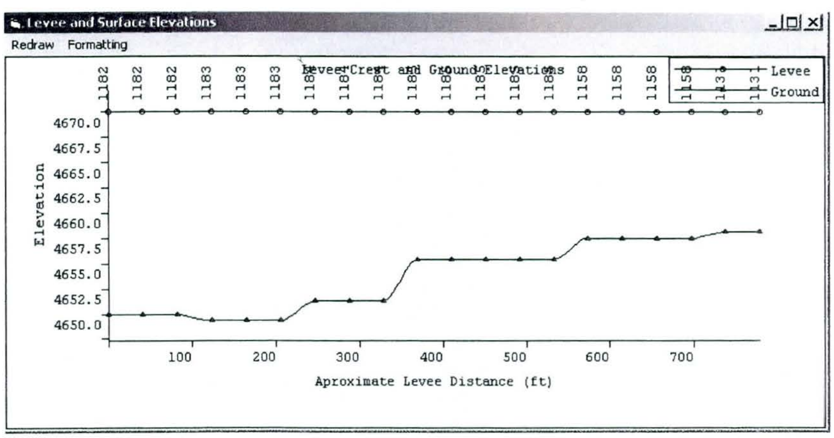
Step 7: Examine the profile of the levee

Select *Levee Profile* from the *Tools* menu to view the levee profile. Then click on the eastern most levee element.

> Note: Use the levee profile plot to verify that the levee elevation and grid element elevation make sense. If the levee is only slightly above the grid element, you may want to make adjustments to one or the other.



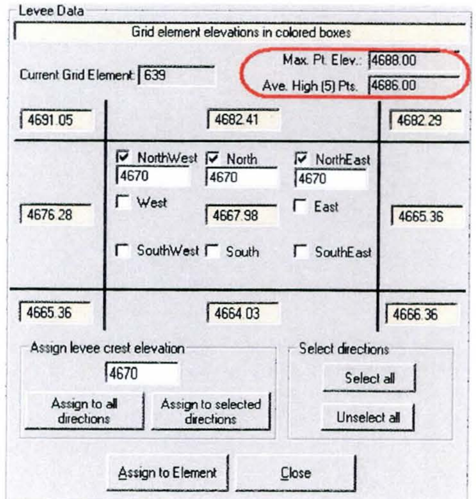
The levee profile plot shows the levee elevation compared to the coinciding grid element elevation. The window shows that the levee has a constant elevation. Use the *Levee Express Editor* to define the levee crest elevation in the next step.



Step 8: Use DTM point data to determine levee crest elevation

Start by importing the *dtm.pts* file from *Lesson 1*. Click the *Levee Express Editor* button. Double click on any grid element that contains a levee. Query individual DTM points using the *Track Elevation Points* button. The elevation of the highest DTM point elevation and the average of the 5 highest points in the grid element is available on the dialog box. Use this data judiciously to define the levee crest elevation.

Note: It is only recommended that this feature is used when the DTM data is dense and accurate. Do not use this feature for walls.

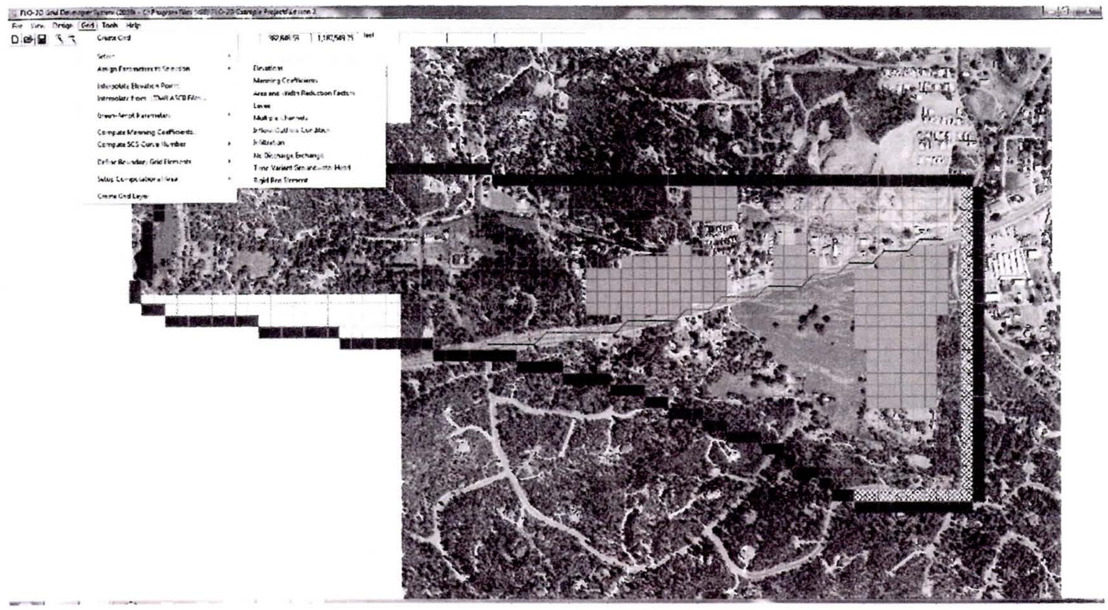


Step 9: Graphically edit infiltration parameters

FLO-2D can simulate infiltration using either the Green-Ampt or SCS curve number methods. In this simple example, set up global infiltration and create several impervious zones. An advanced infiltration tutorial is provided in Lesson 7.

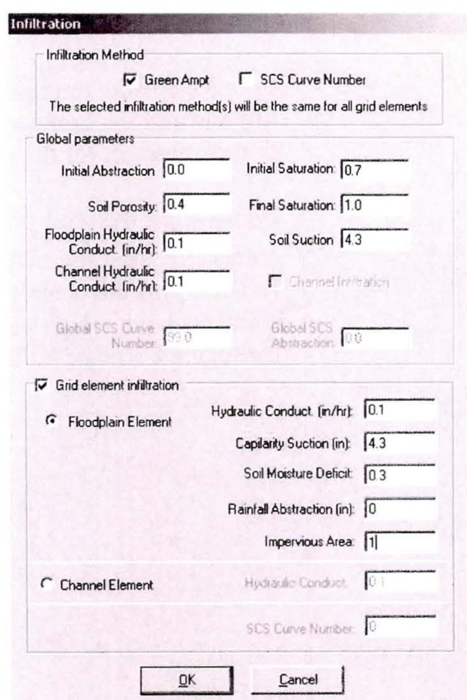
In this step add an impervious surface to populated neighborhoods, parking lots and the highway. First, assign the uniform Green-Ampt infiltration parameters for the project area.

Zoom in on the west portion of the grid system from where the highway enters it. Using the *Select Elements Defined by Polygon* button, select the grid elements that correspond to impervious areas. These grid elements have a high percentage of impervious surfaces. From the *Grid* menu, click *Assign Values to Selection/Infiltration.*



Note that the global infiltration parameters are already set to default values and the default infiltration method is Green Ampt. Adjust the global infiltration parameters at this time.

To assign the individually selected impervious grid elements, enter 1 in the *Impervious Area* dialog box for the *Floodplain Element.*



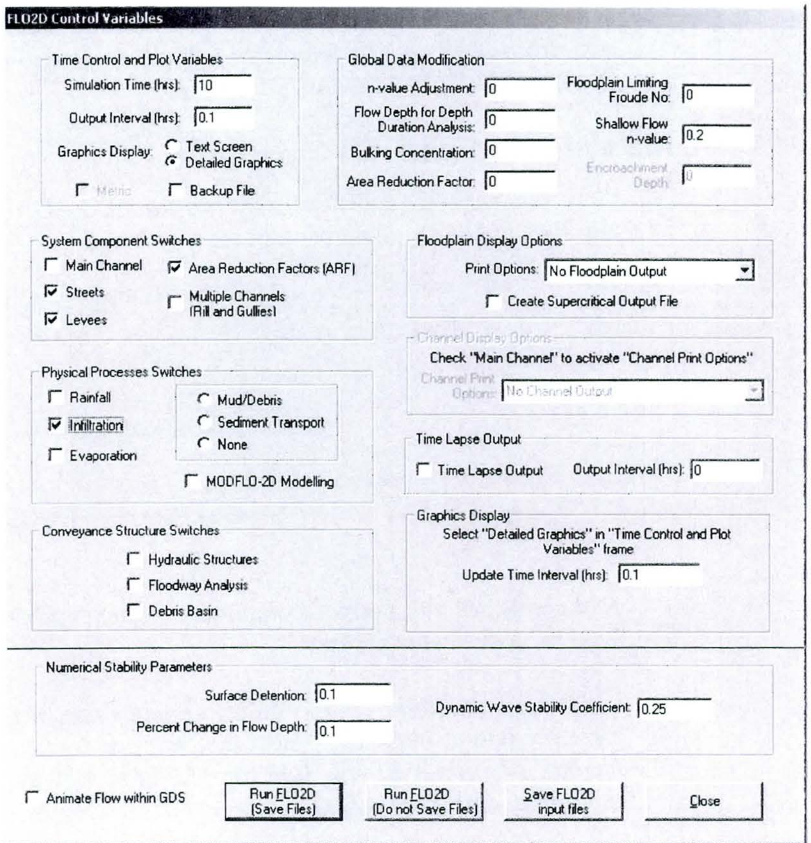
Click *OK* and notice that the local infiltration areas are symbolized by red cross-hatched grid elements. Hide the local infiltration grids by clicking *View/Components/Infiltration.*

Step 10: Save the data files and run the model

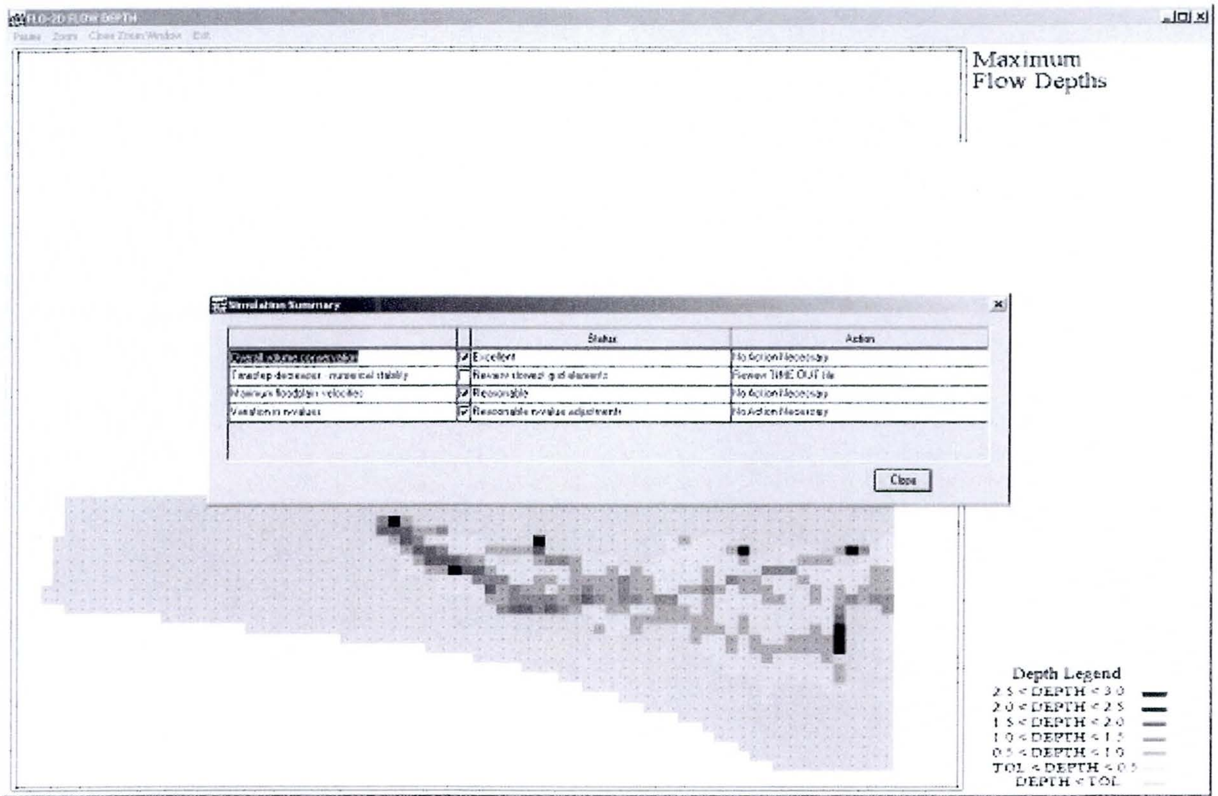
Save the FLO-2D data files. To run the FLO-2D flood simulation, use the *Run FLO-2D…* command in the *File* menu.

Make sure all the correct component switches are turned on (Streets, Levees, ARF and Infiltration).

Input the *Simulation Time* = 24 hr, *Output Interval* 0.25 hr (or 0.10 hrs for more output intervals), select the *Detailed Graphics* option and input *Update Time Interval* = 0.10 hr (or 0.05 hr to view the flood progression more frequently) as shown.



Now click *Run FLO-2D (Save Files)* and monitor the floodwave progression over the project area. The inflow hydrograph will be plotted and the volume conservation is reported during the simulation.



Summary

In this lesson, the GDS was used to create and edit graphical features. When building a detailed urban flood model, each grid element can be modified to represent a specific floodplain feature such as a building, street, obstruction, infiltration or levee. Each feature can affect the flood distribution on the grid system. For large flood events, a lot of floodplain detail may not be necessary as the flooding will cover the much of the available floodplain anyway. For small flood events, more detail can enhance the flood map resolution.

Review Questions

1. When it is necessary to revise grid element elevations?

   ____

   ____

   ____

   ____

2. Streets can perform in the model as which of the following? Check all that apply.

   - ☐ Conveyance channels
   - ☐ Embankments
   - ☐ Obstructions
   - ☐ Storage loss features

3. Levees can function as which of the following? Check all that apply.

   - ☐ Berms
   - ☐ Railroad embankments
   - ☐ Floodwalls
   - ☐ Storage loss features
   - ☐ Conveyance channels
   - ☐ Detention basin dams

4. Which of the following are most likely to be assigned ARF values?

   - ☐ Warehouse
   - ☐ Mobile homes
   - ☐ Beach houses on piles
   - ☐ Parking garage
   - ☐ Levees
   - ☐ Baseball stadiums

5. What methods are provided to compute spatially variable infiltration?

   ____

6. Street global parameter assignments such as curb height will override the individual grid element street parameters? (True or False)

7. Which of the following will directly affect grid element flood storage?

   - ☐ Streets
   - ☐ Infiltration
   - ☐ Buildings
   - ☐ Levees

8. Floodplain details such as streets and buildings are more important to define the area of inundation for shallow flow or deep flooding? ____________________

LESSON 3 – GDS CHANNEL FROM SCRATCH

Overview

Channel routing is an essential tool in a FLO-2D project. The channel component should be used when the defined channel geometry will convey a significant portion of the flood volume. Channel flow is simulated as 1-D with defined cross sections. The channel component is used to simulate channel-floodplain exchange including return flow to the channel. This lesson will demonstrate how to create a channel from scratch using the GDS by generating a channel segment, editing channel parameters and modifying a channel segment.

Required Data

The six basic data files for FLO-2D floodplain model are provided for this lesson. The files required to create a new channel are.

- CONT.DAT
- TOLER.DAT
- FPLAIN.DAT
- CADPTS.DAT
- INFLOW.DAT (optional)
- OUTFLOW.DAT (optional)

Aerial imagery is an important tool to locate the channel with respect to the grid system.

File	Content	Location*
**.dat*	Grid System Data File	*Example Projects\Lesson 3*
Goat.tif	Aerial photographs	*Example Projects\Lesson 1*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To build a channel in the GDS these steps should be followed.

- Open the Grid Developer System (GDS) program;
- Load the model *Lesson 3*;
- Import aerial images;
- Create a channel segment;
- Realign the channel;
- Edit channel segment parameters;
- View and edit bed slope;
- Input a channel inflow and outflow node;
- Save project and run the FLO-2D model;
- Confluence and NOFLOC's.

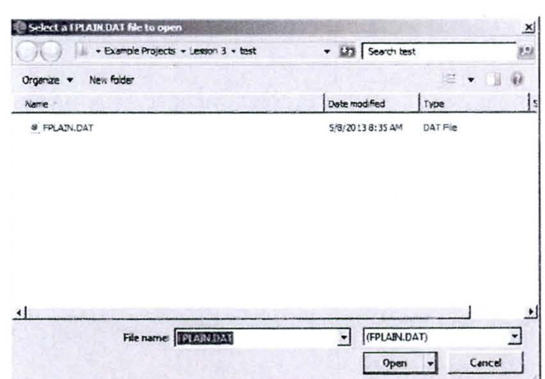
Step 1: Open the Grid Developer System program

Double click the GDS Basic icon from the desktop.

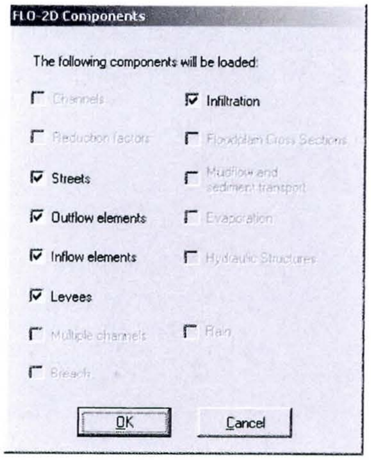
Step 2: Load the model Lesson 3

Click the *File* menu in the GDS and select the *Open Existing FLO-2D Project* command on the File menu.

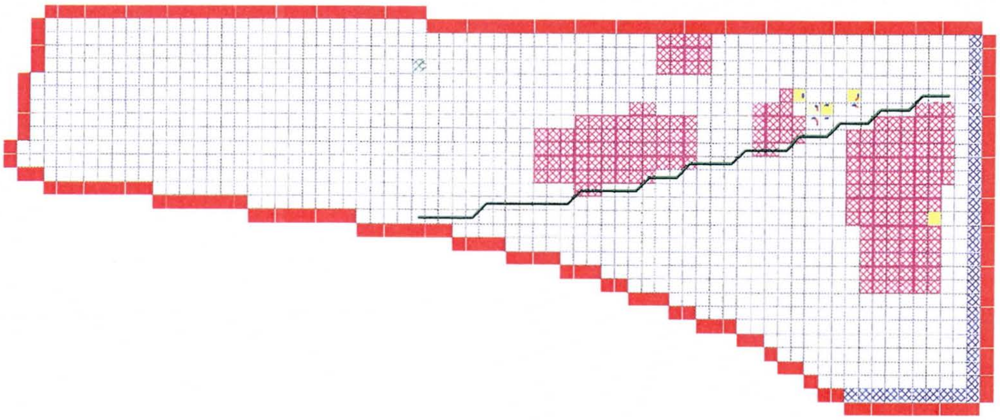
Browse to the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation Example Projects\Lesson 3* and double click the file *FPLAIN.DAT*.



The following dialog box will be displayed. Click *OK* to load the model and display the grid system.



The grid system should look like this.

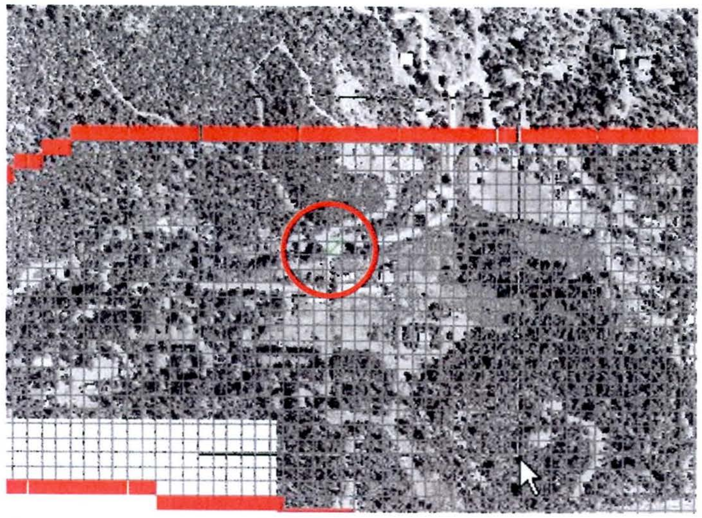


Turn off the infiltration graphics by clicking *View/Components /Infiltration*.

Step 3: Import aerial images (see Lesson 1 – Step 4)

Step 4: Create channel segment

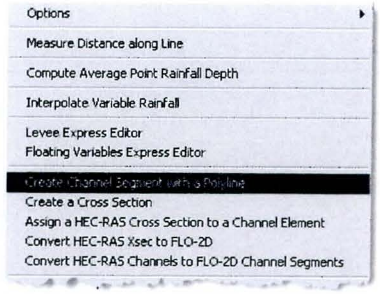
Start by zooming-in on the upper left hand quadrant of the grid system.



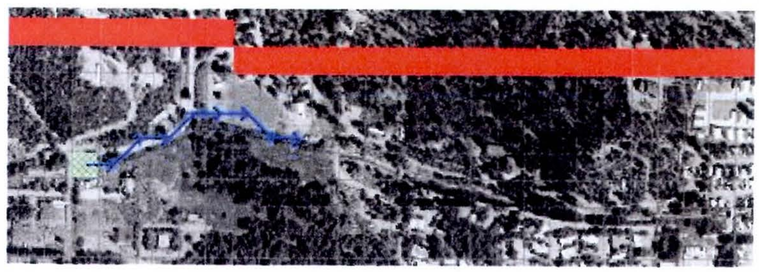
Locate the natural channel on the aerial photo by zooming in on the floodplain inflow node that is in the upper left quadrant. The dotted line in the following figure shows the location of Goat Camp Creek.

Note: The floodplain inflow node may not correspond to the channel. You can reassign the inflow node to the channel later.

Select *Create Channel Segment with a Polyline* from the *Tools* menu.



Start the channel by clicking on the *inflow node* and then clicking along the channel in the image. The channel elements are automatically filled as click along the path of the polyline.



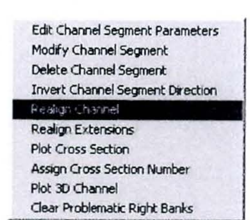
Double click the last channel element to accept the channel. As the cursor moves to the edge of the screen, the grid system will automatically scroll in the direction of the cursor.

Note: If the autoscroll is too fast, zoom out and try again.

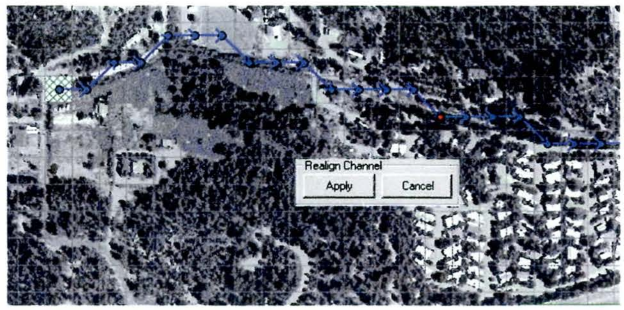
Step 5: Realign the channel

In order to realign the channel, click on the channel and click *Realign Channel*.

Note: Realign the channel before editing the geometry data.

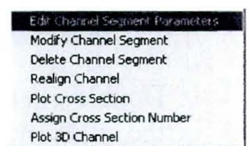


The channel feature will show dots in the middle of each channel grid element. Click and drag the dot to the new channel element. Click *Apply* to accept changes.

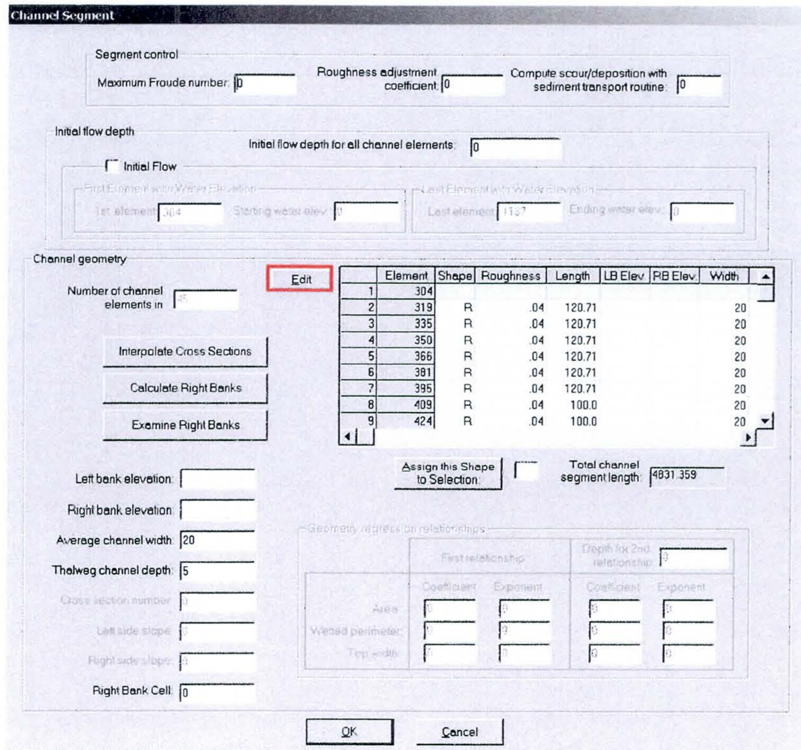


Step 6: Edit channel geometry parameters

Once the channel delineation is complete, define the channel geometry parameters. Click on any channel element within the segment to access the editor dialog box.



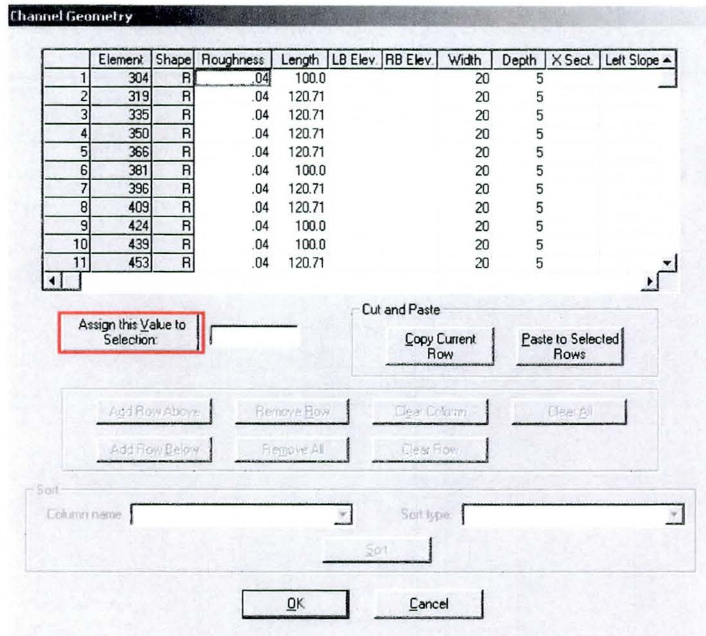
Select *Edit Channel Segment Parameters* to display the *Channel Segment* dialog box. Please note that there are data dependencies and the grayed data will be turned on with the appropriate data selection. Refer to the data Input manual for channel variable definitions.



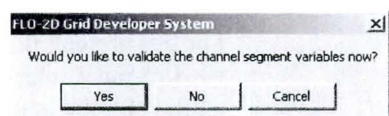
For this project, the channel geometry will be defined as a rectangle. Enter a channel width and depth for each channel element. Click the *Edit* button [Edit] in the *Channel Geometry* group of the dialog box.

Use the *Channel Geometry* dialog box to enter the channel parameters. For example, enter a rectangular channel depth of 5 ft and a width of 20 ft. To enter multiple values, simply highlight all of the channel element boxes, and assign a value to the data input box to the left of the *Assign this Value to Selection* [Assign this Value to Selection:] button and click on the button.

Note: The channel element number can only appear once in the CHAN.DAT file. To create a channel confluence with a tributary channel it is necessary to locate the last tributary channel element contiguous to a main channel in one of the 8 flow directions.



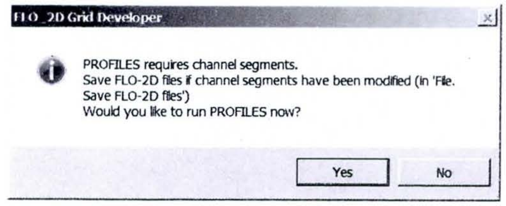
Click *OK* to close the *Channel Geometry* dialog box and click *Yes* to validate the channel data entries and close the *Channel Segment* dialog box.



Step 7: View and edit bed slope

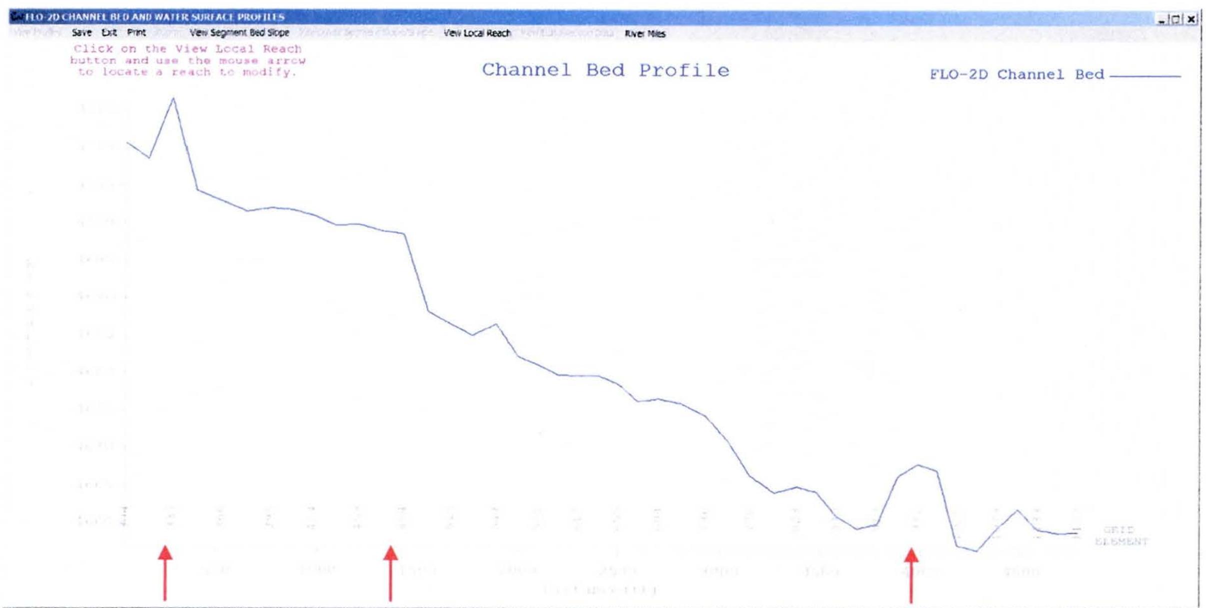
Click *File/Save FLO-2D Files...* Save the files in a test folder. Click *File/Run PROFILES*.

Click *Yes* to run *PROFILES*.

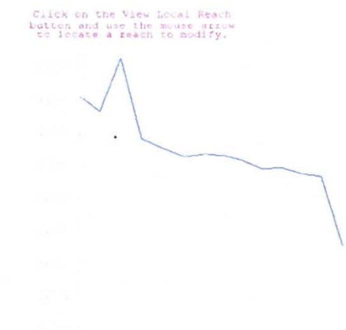


On the *PROFILES* window, click *View Segment Bed Slope*.

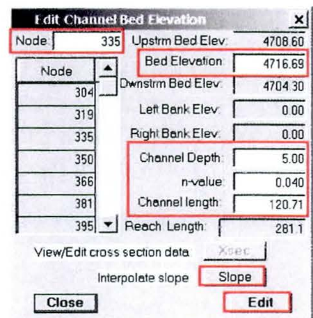
The bedslope is shown in the following figure. The image shows that the bedslope is not always positive and becomes fairly steep. The spikes in elevation are caused by the higher elevation of the grid element associated with the channel. The red arrows show areas of concern that can be edited in the PROFILES program. The arrows are graphics and will not show up in PROFILES.



Click *View Local Reach* and see that the cursor has changed to a vertical arrow. Align the arrow with the first spike and click the left mouse button.

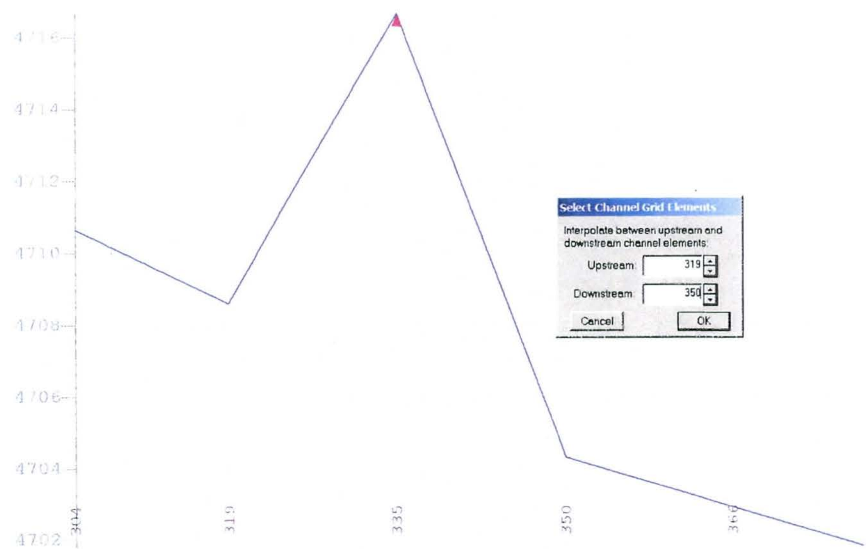


Click *View/Edit Xsection Data* to bring up the following dialog box. The white boxes are values that can be edited in the *PROFILES* program. Editing the Bed Elevation of the channel in this box will result in edited grid element elevation in the fplain.dat file for grid element 335.



Manually edit the slope by entering a value in the *Bed Elevation* box and clicking *Edit* or linearly interpolate the slope between 2 grid elements by clicking the *Slope* button.

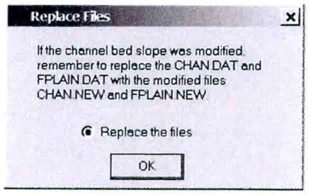
In the interpolation dialog box Enter the Upstream element and the Downstream element and click interpolate.



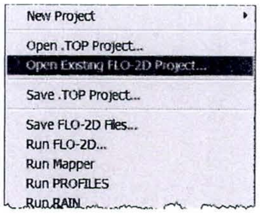
Repeat this procedure until the channel bed slope profile has no more spikes or steep elevations. *Pan* along the zoomed channel using the *UP* and *DOWN* arrows on the keyboard.

The finished channel bedslope has no spikes or steep slopes and the final channel element is lower than the contiguous upstream element.

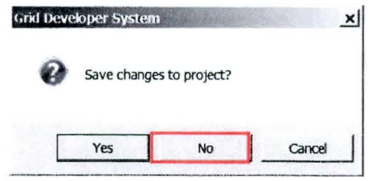
Click the *Save* button and select the *Replace the files* radio button to save the data changes and save over the existing *chan.dat* and *fplain.dat* files. Click *Exit* to close the *PROFILES* program.



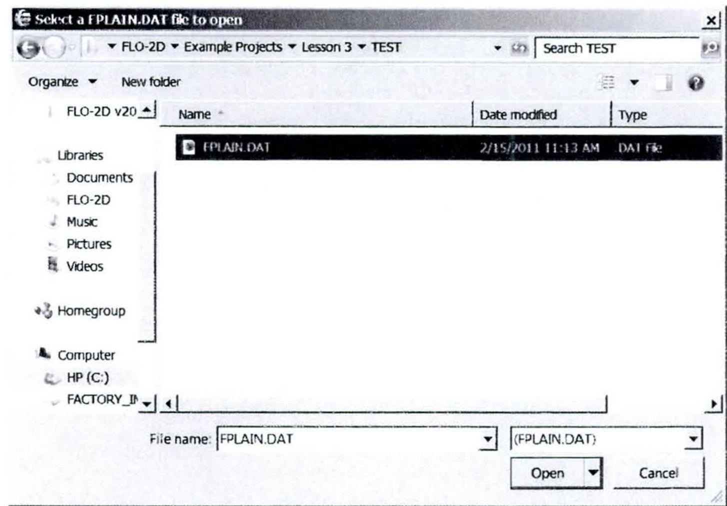
The data files were edited outside the *GDS* so it is necessary to reload the project in the *GDS*. On the *GDS* window, click *File/Open Existing FLO-2D Project...*



Do not save the project. Click *No*

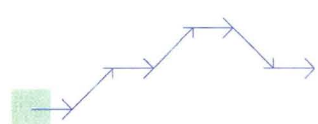


Navigate to the project folder and click *Open*.

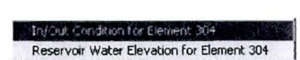


Step 8: Input a channel inflow

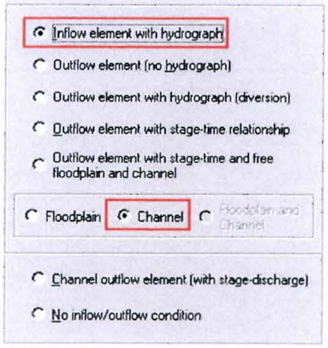
The channels can have multiple inflow hydrographs at various locations. For this project, re-assign the floodplain inflow hydrograph to a channel node. Right click on the first channel element of the segment and click *In/Out Condition for Element *****.



NOTE: Your element may not be #304.



Change the inflow node from a floodplain node to a channel node by clicking the *Channel* radio button and click *OK*.

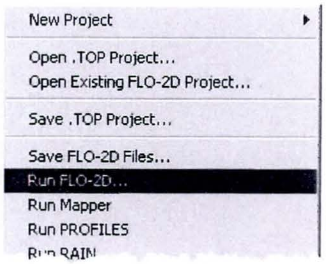


Step 9: Save project and run the FLO-2D flood simulation

To Save the project click the *Save FLO-2D Files…* command in the *File* menu.

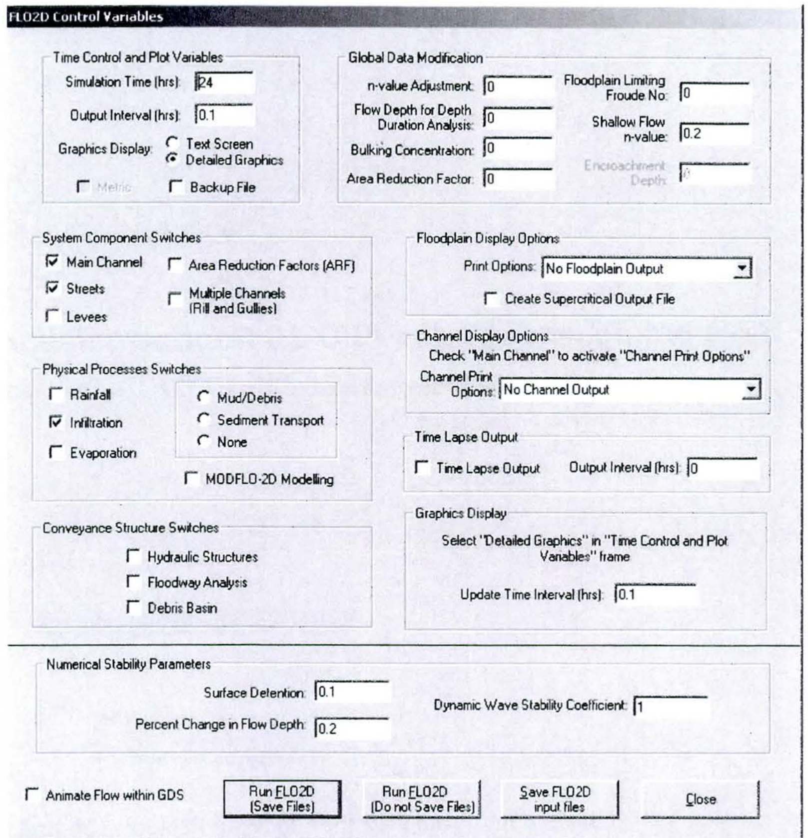
Create a new folder in the Example Projects subdirectory called *Channel Test* (or any name) and save the data files. Avoid overwriting the data files of other Example Projects in order to use them later.

To run the FLO-2D flood simulation, use the *Run FLO-2D…* command in the *File* menu.



The main control dialog box will. Note the *Main Channel* box has been checked in the *System Component Switches Group*. This switch enables the channel flow to be simulated. Input a *Simulation Time* = 24 hr, *Output Interval* 0.25 hr (or 0.10 hrs for more output intervals), select the *De-*

tailed Graphics option and input *Update Time Interval* = 0.25 hr (or 0.05 hr to view the flood progression more frequently) as shown.



Now click *Run FLO2D* and observe the flood progressing down the channel and overtopping the banks. The volume conservation and inflow hydrograph can also be monitored.

Step 10: Create a channel confluence and define NOFLOC's

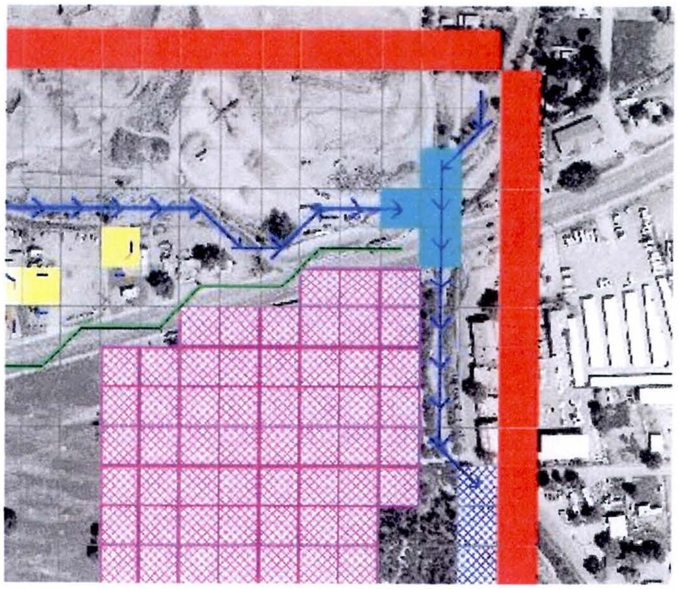
Channel confluences (or split flow channel) require channel elements as neighbors in three or more flow directions. The model searches all eight flow directions for potential contiguous channel elements.

Before creating the new channel, delete all outflow nodes on the northwest half of the grid system. Zoom-in on the top right corner of the grid system. There is a creek running north to south. *Create a channel* for this creek that has a confluence near the highway. *Assign the channel geometry* similar to the method used in *Step 6*. Once the new segment is created, the last channel element of the original channel will be a neighbor to one or more of the new channel elements.

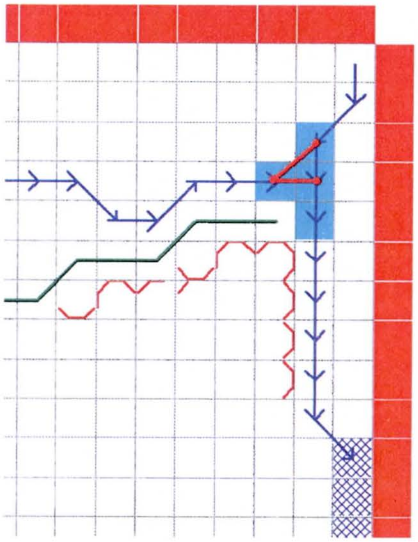
A careful examination of the channel elements will reveal that there are some channel elements that are neighbors but should not share discharge. These channel elements must be identified as 'no flow sharing channel elements' or NOFLOCs. NOFLOCs are defined as channel element that are contiguous but will not share discharge during the flood simulation.

The GDS will automatically identify all the potential NOFLOC pairs that may be required.

Note: Each channel grid element should only be listed once in the CHAN.DAT file.

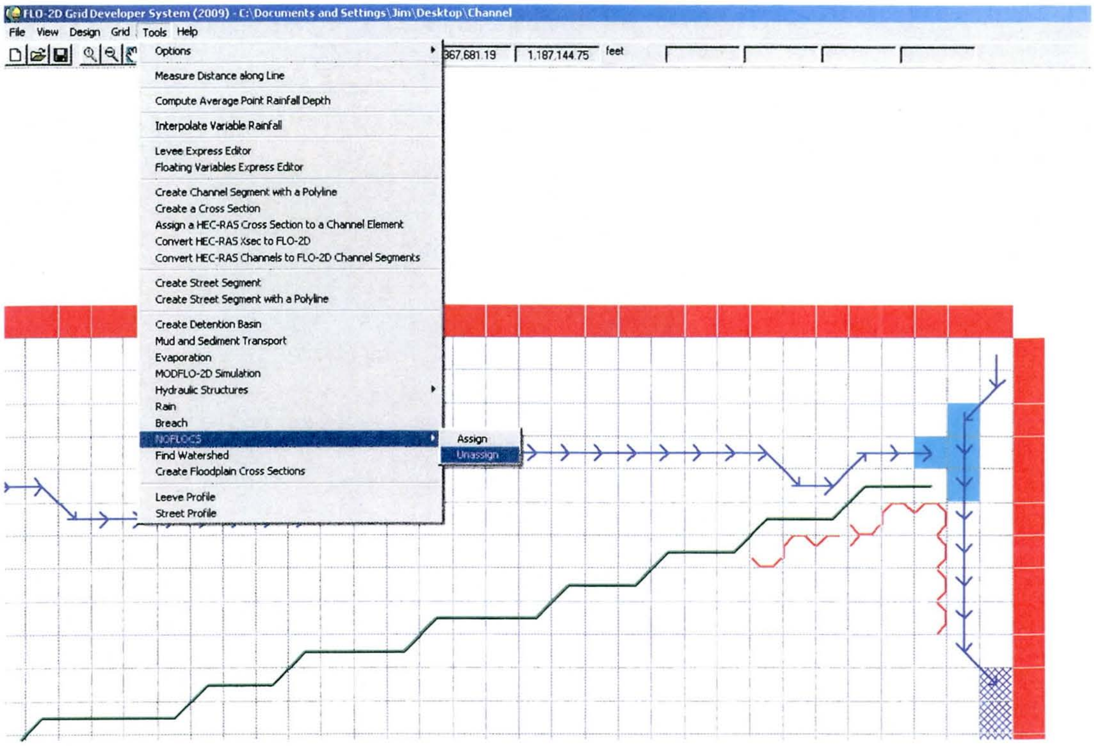


The red lines in the following image indicate that two pairs of channel elements (one channel element is common to both pairs) that should not share discharge and need a NOFLOC assigned.

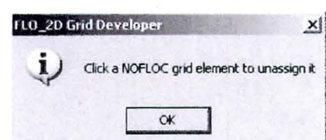


Note: The red lines connecting the noflocs will not be seen in the GDS. They are added to the image graphically.

To make adjustments to the NOFLOC assignment, click *Tools/NOFLOCS/Unassign*.

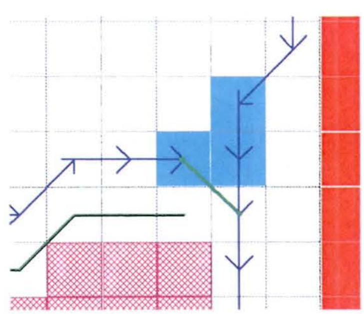


Click OK on the following dialog box and then click the unnecessary NOFLOC channel element to remove it.



The green line shows the two channel elements that can now share discharge. The green line is not shown in the GDS. It was graphically added to the image.

Note: The NOFLOC list is found at the end of the CHAN.DAT file. Review this list if you need to make sure that the NOFLOC pairs have been correctly assigned.

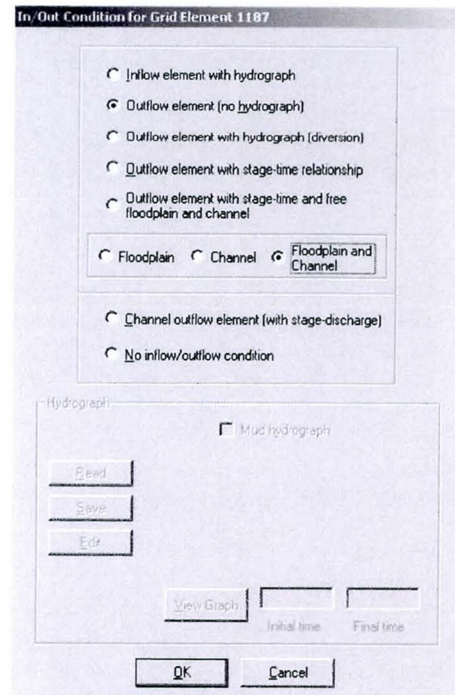


Step 11: Create a channel outflow node

For the channel outflow node, right click on the last channel element of the 2nd channel segment. Click the *In/Out Condition for Element* **** button.

NOTE: Your element may not be #4759.

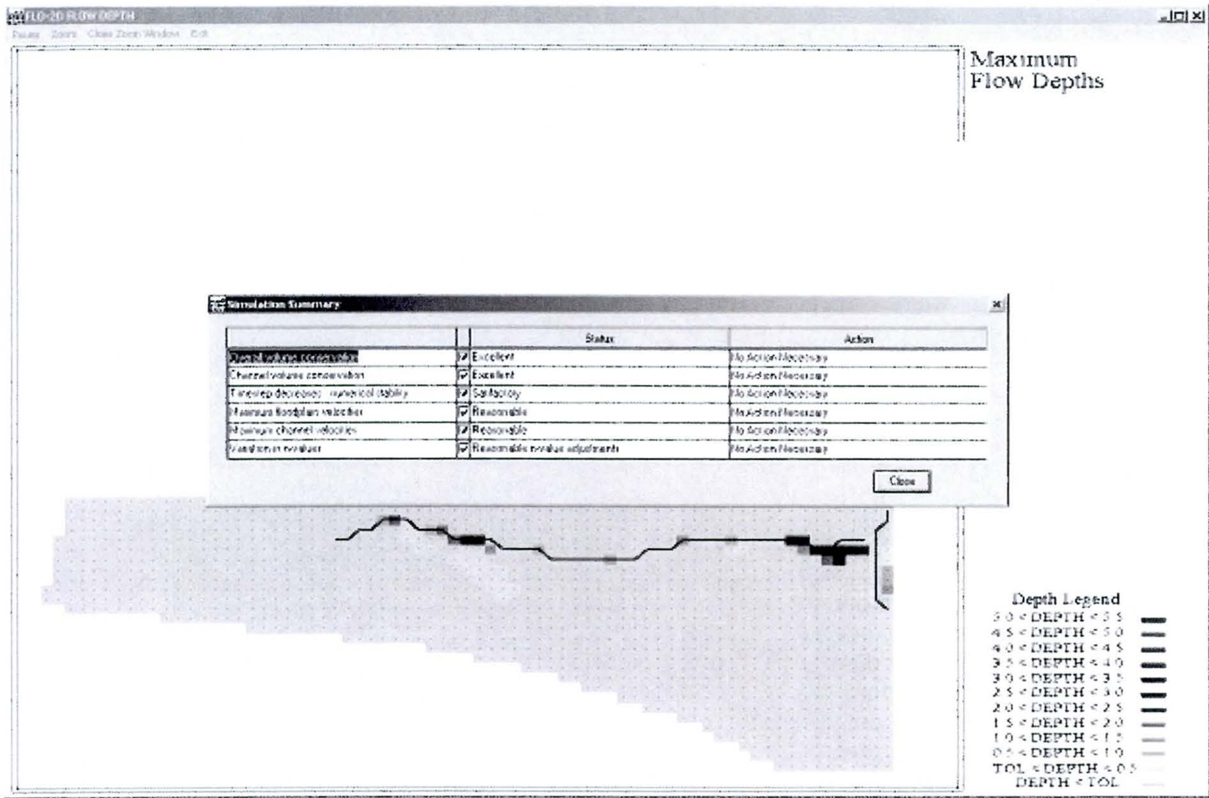
Fill in the dialog box as shown below and click *OK*.



Note: By assigning a channel outflow node with no hydrograph, it is assumed that the channel outflow discharge will be computed as normal depth. The inflow to the channel outflow will be discharged off the grid system using an estimate of normal flow depth.

The elevation of the channel thalweg associated with a channel outflow node must be lower than that of the upstream node. When the channel geometry is set to rectangular or trapezoidal, the GDS assigned the thalweg elevation relative to the grid element elevation. In order to ensure the thalweg elevation of the outflow node is lower than the upstream node, it may be necessary to lower the grid element elevation of the outflow node.

Step 12: Save and run the model



Summary

This example project covered the steps to build a simple rectangular channel. It demonstrates the ease of creating channels with the GDS. There are additional channel defining tools available in the GDS. For Example, the GDS can also import and interpolate natural channel geometry. It is possible to import channel profiles and cross sections from HEC-RAS and define bank elements elevations and locations.

Review Questions

1. Describe 2 ways to set the bank element elevation for a rectangular channel. Which is default?

____

____

____

2. Two-dimensional channel flow in FLO-2D is simulated in an algorithm similar to the HEC-RAS unsteady flow model? (True or False) ________________

3. Describe 3 methods to edit the bedslope of a channel? ____________________

____

____

____

4. What are the four types of channel cross section used in the FLO-2D model?

____

____

____

____

5. The XLEN variable in the CHAN.DAT file represents which of the following?
   - ☐ Channel length between cross sections
   - ☐ Channel top width
   - ☐ Channel length inside the grid element
   - ☐ Distance between the centers of channel nodes
6. Each channel must have an inflow and outflow node? (True or False) ______________
7. A description of NOFLOC assignment can be found where? Check all that apply.
   - ☐ Data Input Manual Chapter 2
   - ☐ Basic Channel webinar video
   - ☐ Channel PowerPoint presentation in the FLO-2D Help folder.
8. The channel default n-value 0.04 is reasonable? (True or False) ______________
9. What are NOFLOC's? ________________________________

____

10. When is it necessary to unassign a NOFLOC?

____

____

LESSON 4 – HEC-RAS CROSS SECTION CONVERSION

Overview

Many FLO-2D projects require improving existing flood hazard maps by using an integrated channel and floodplain flood routing analysis. For some of these projects, the HEC-RAS cross sections will be used to construct the channel data base. This lesson will show how to automatically create a FLO-2D cross section file (XSEC.DAT) from an existing HEC-RAS file. This is straightforward process that involves locating the original HEC-RAS geometry data file and converting it to the XSEC.DAT file.

Required Data

A HEC-RAS cross section file will also be needed for this lesson.

File	Content	Location*
Houston.g06	HEC-RAS Cross Section File	*Example Projects\Lesson 4*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To convert HEC-RAS cross section data to an XSEC.DAT file using the GDS, follow these steps:

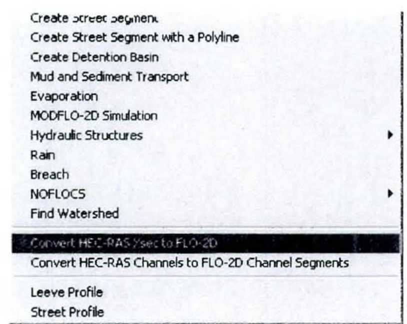
- Open the Grid Developer System (GDS) program;
- Access the cross section conversion application;
- Load the HEC-RAS cross section file;
- Save the XSEC.DAT file;
- Verify the new cross section data.

Step 1: Open the Grid Developer System program

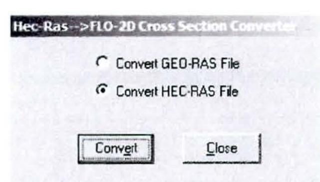
Double click the GDS Basic icon from the desktop.

Step 2: Convert HEC-RAS Cross Section

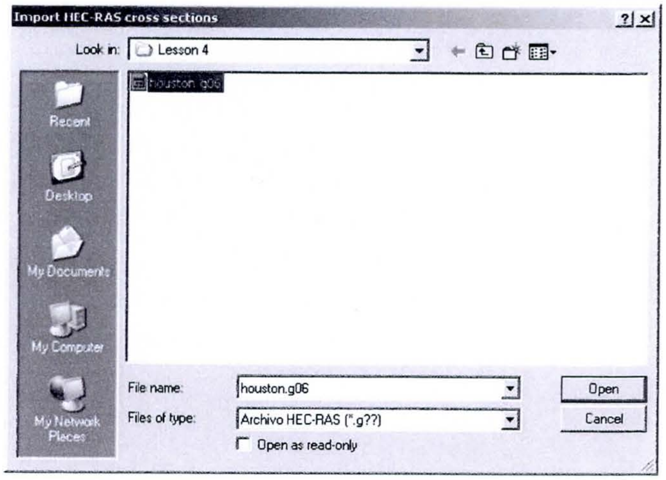
The CHAN.DAT file does not have to exist to create the XSEC.DAT file from HEC-RAS file using the conversion tool. Click on *'Convert HEC-RAS Xsec to FLO-2D'* command under the *'Tools'* menu.



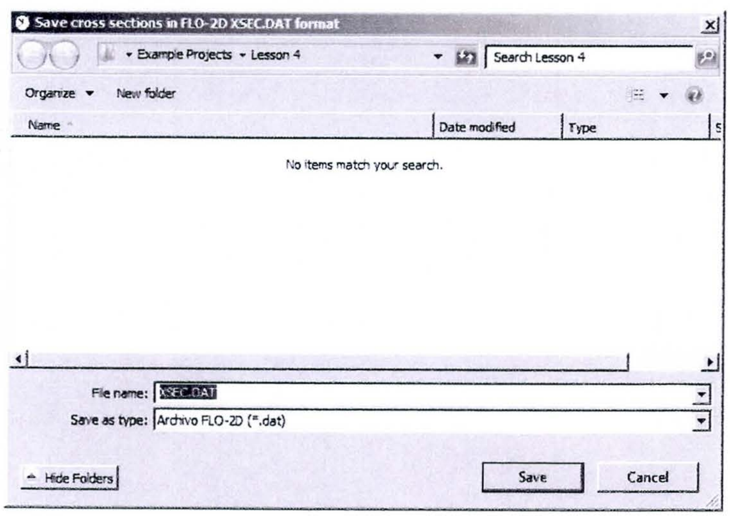
A dialog box to initiate the conversion will appear. Click on the type of HEC-RAS file that is to be converted. In this case it is a typical HEC-RAS file with a *.g?? extension.



Click on the *'Convert'* button to locate the HEC-RAS file (Houston.g06) to be converted. This file located in the Lesson 4 subdirectory along with the other project data files.



Click on the *'Open'* button to display a second dialog box to select the FLO-2D data file name to be saved.



WARNING: The HEC-RAS cross sections are probably organized from downstream to upsteam. If this is the case, it will be necessary to rewrite the XSEC.DAT from upstream to downstream.

Enter the File Name: *'XSEC.DAT'* and click the *'Save'* button and the following dialog appears confirming the conversion to the FLO-2D data file format is completed.

This completes the HEC-RAS cross section data file conversion process.

Step 3: Validate Cross Section Data

Open the *XSEC.DAT* file with any text editor such as WordPad© or UltraEdit©. The cross section data will look something like the following:

```
X 19
1000.00 4670.00
1200.00 4666.00
1300.00 4664.00
```

155.04 664.00
1590.00 4660.00
1600.00 4659.00
1610.00 4660.00
1635.00 4668.00
1665.00 4670.00

Each cross section begins with an identification character 'X' and a cross section number '19' on the first line. Subsequent lines contain station and elevation data. Some of the cross sections are missing a cross section name. The conversion program did not print the name on all cross sections because it was missing from the HEC-RAS data. The GDS Pro uses the Rivermile instead of the description so all cross sections get a name.

Check that each cross section for the appropriate data. For example, the HEC-RAS data may not be spaced uniformly because it is missing a decimal place. This can cause the conversion to shift a decimal place. Make sure the station data in column 1 is always increasing. In the following corrected data file, see that the name was added in line 1 and the station (1550) and elevation data was corrected as underlined below:

NOTE: The program may not need all the HEC-RAS cross sections that were written to the XSEC.DAT. Review the XSEC.DAT and delete any unnecessary cross sections.

X 19 Goat19
1000.00 4670.00
1200.00 4666.00
1300.00 4664.00
1550.00 4664.00
1590.00 4660.00
1600.00 4659.00
1610.00 4660.00
1635.00 4668.00
1665.00 4670.00

Summary

This lesson demonstrated how to extract and convert HEC-RAS cross sections to the FLO-2D XSEC.DAT file. Once this process is complete review the XSEC.DAT file, remove any unnecessary cross sections and proceed to locate the cross sections in the GDS with respect to the project channel. Then in the GDS outline the channel and perform the cross section interpolation. It may also be necessary to review the cross sections to ensure that they have appropriate top of bank elevations.

Review Questions

1. True or False? Each channel element must have a surveyed cross section? ___________
2. True or False? FLO-2D will use all the HEC-RAS cross sections? ________________
3. Which portion of the HEC-RAS cross section does the FLO-2D model use?

   ____

4. True or false? FLO-2D requires a name for each cross section? ___________________
5. True or false? HEC-RAS cross sections may be spaced closer together than the FLO-2D grid element spacing? ____________
6. True or false? HEC-RAS cross sections may be spaced farther apart than the FLO-2D grid element spacing? _____________
7. The HEC-RAS cross sections are primarily important in the FLO-2D model for which of the following?
   - ☐ Channel length inside the grid element
   - ☐ Channel cross section flow area
   - ☐ Channel n-value
   - ☐ Channel energy slope
   - ☐ Channel surface area
8. Complete this statement: "The HEC-RAS cross section data must be edited if the flood-plain elevations are…" ____

   ____

9. True or False? If the HEC-RAS cross sections are separated by more than one grid element, the cross section data must be interpolated? _______________

LESSON 5 – PROFILES CROSS SECTION INTERPOLATION

Note: In GDS 2009 the cross section interpolation is automated. It is no longer necessary to perform the initial interpolation as described in this workshop. It is still possible to perform those steps and that is why they are presented here. It is also possible to perform interpolations between one or more cross sections and that is described in below the automatic interpolation steps.

Overview

In most river-reach data bases, there is only one surveyed cross section for every five to ten channel elements. It is necessary therefore to interpolate cross section elevation and geometry for those channel elements between the surveyed cross sections. Between two surveyed cross sections, the channel geometry is variable and thus interpolating between the surveyed cross sections enables the channel elements to have a unique cross section shape. This lesson shows how to use the PROFILES program to interpolate cross section data and link it to the channel element data.

Required Data

The profiles program requires the six basic data files plus two channel files. These files already exist and are found in the folder *Lesson 5*.

- CONT.DAT
- CADPTS.DAT
- FPLAIN.DAT
- TOLER.DAT
- INFLOW.DAT (optional)
- OUTFLOW.DAT (optional)
- CHAN.DAT
- XSEC.DAT

File	Content	Location*
**.dat*	Grid System Data Files	*\Example Projects\Lesson 5*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To interpolate cross section data using the PROFILES program follow these steps.

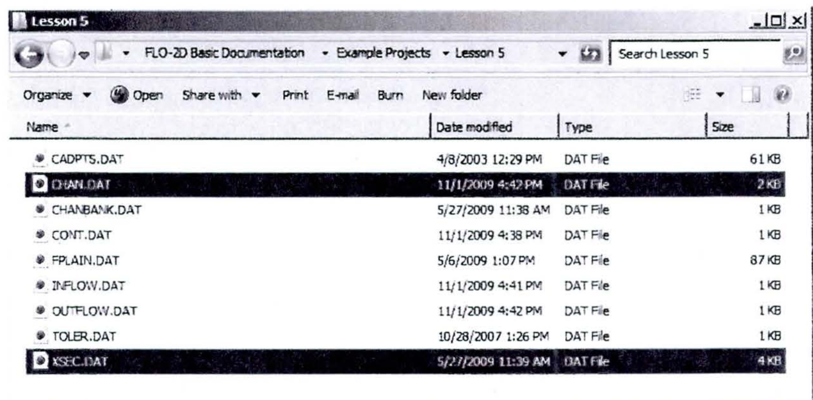
- Create a new project folder
- Open the FLO-2D GDS and load the *Lesson 5* project;
- Examine pre-interpolation channel data files;
- Open the PROFILES program;
- Automatically interpolate all cross section data;
- Examine the post-interpolation channel data files;
- Edit a cross section;
- Manually interpolate between two cross sections.

Step 1: Create a new project folder

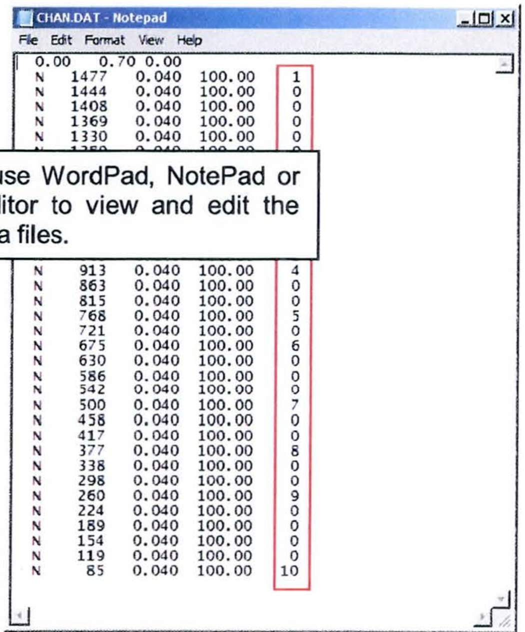
In Explorer, copy the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation Example Projects\Lesson 5* and rename it to *Lesson 5.1*. If irreparable mistakes are made, start over with the *Lesson 5.1* folder.

Step 2: Open the data files in NotePad

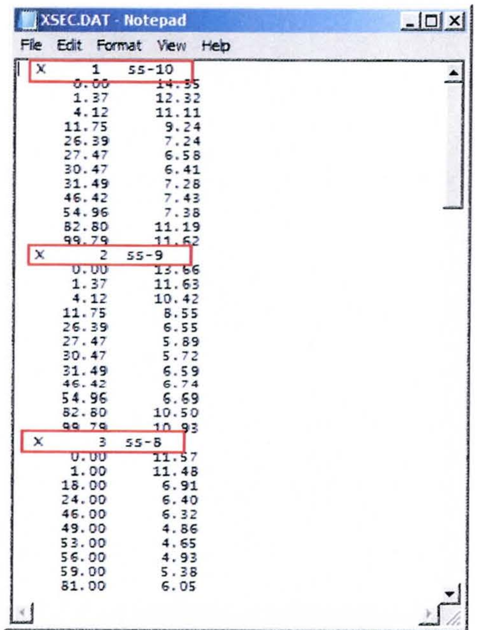
Browse to the folder *Lesson 5* and open CHAN.DAT and XSEC.DAT in NotePad.



Step 3: Examine the pre-interpolation channel data files


CHAN.DAT - Notepad

```
0.00     0.70 0.00
N   1477     0.040   100.00     1
N   1444     0.040   100.00     0
N   1408     0.040   100.00     0
N   1369     0.040   100.00     0
N   1330     0.040   100.00     0
N    913     0.040   100.00     4
N    863     0.040   100.00     0
N    815     0.040   100.00     0
N    768     0.040   100.00     5
N    721     0.040   100.00     0
N    675     0.040   100.00     6
N    630     0.040   100.00     0
N    586     0.040   100.00     0
N    542     0.040   100.00     0
N    500     0.040   100.00     7
N    458     0.040   100.00     0
N    417     0.040   100.00     0
N    377     0.040   100.00     8
N    338     0.040   100.00     0
N    298     0.040   100.00     0
N    260     0.040   100.00     9
N    224     0.040   100.00     0
N    189     0.040   100.00     0
N    154     0.040   100.00     0
N    119     0.040   100.00     0
N     85     0.040   100.00    10
```


XSEC.DAT - Notepad

```
X        1    55-10
     0.00       14.35
     1.37       12.32
     4.12       11.11
    11.75        9.24
    26.39        7.24
    27.47        6.58
    30.47        6.41
    31.49        7.28
    46.42        7.43
    54.96        7.38
    82.80       11.19
    99.79       11.62
X        2    55-9
     0.00       13.66
     1.37       11.63
     4.12       10.42
    11.75        8.55
    26.39        6.55
    27.47        5.89
    30.47        5.72
    31.49        6.59
    46.42        6.74
    54.96        6.69
    82.80       10.50
    99.79       10.93
X        3    55-8
     0.00       11.57
     1.00       11.48
    18.00        6.91
    24.00        6.40
    46.00        6.32
    49.00        4.86
    53.00        4.65
    56.00        4.93
    59.00        5.38
    81.00        6.05
```

Note: The use WordPad, NotePad or any text editor to view and edit the FLO-2D data files.

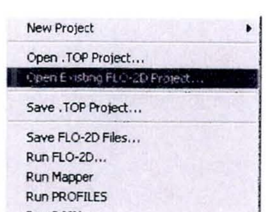
The 0's in the last column of the *CHAN.DAT* file represent channel elements that do not have an assigned surveyed cross section. In the following steps, an interpolated cross section is created for each channel element and replace the 0's.

Warning: Natural river cross sections should have at least 10 points to define the channel geometry. If there are less than 6 points, the interpolation will fail.

In the XSEC.DAT file, there are only 10 surveyed cross sections. These are assigned to the CHAN.DAT in the last column. Assignments are based on the geographical location of the cross section coordinates in proximity to the channel element coordinates. The first and last channel elements in a segment should have a surveyed or cut cross section.

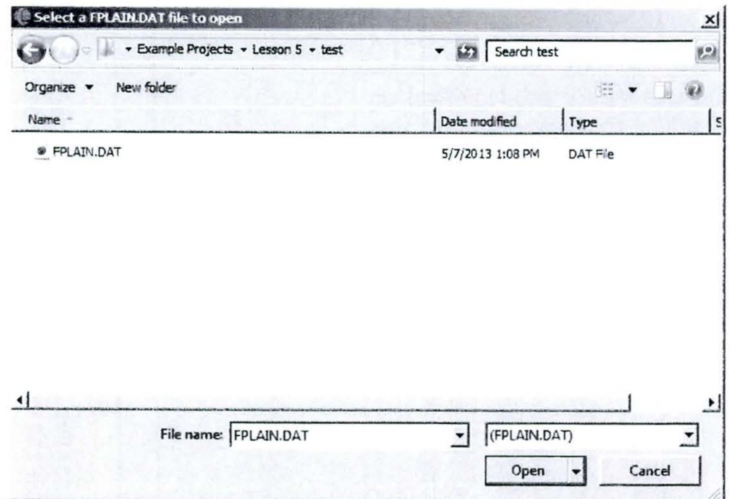
Step 4: Load the Project in the GDS and run the PROFILES program

Double click the GDS Basic icon from the desktop.

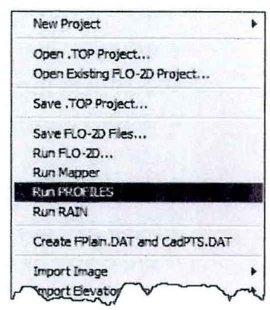


Click the *File* menu in the GDS and select the *Open Existing FLO-2D Project* command on the File menu.

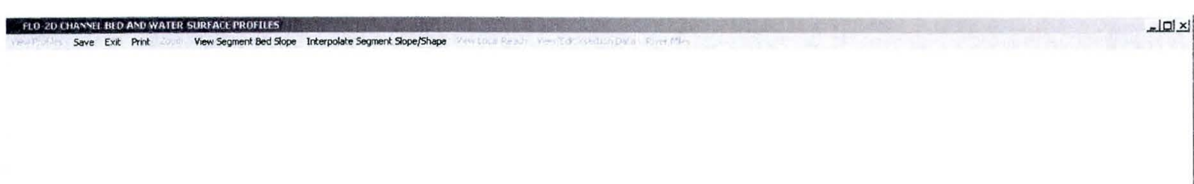
Browse to the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation Example Projects\Lesson 5* and double click the file *FPLAIN.DAT.*



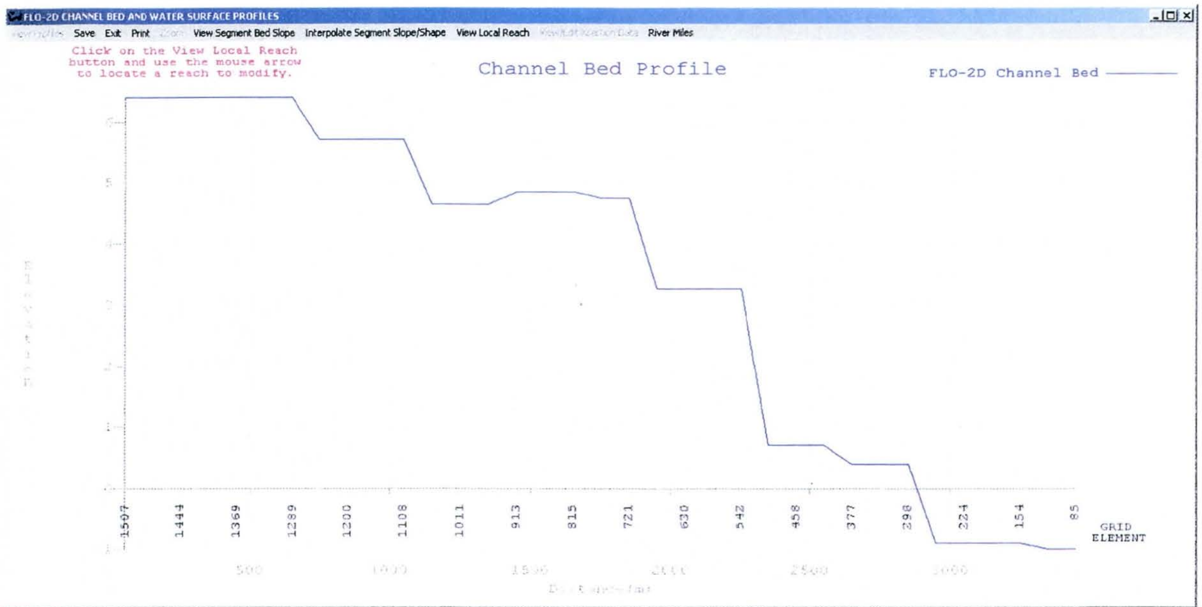
From the GDS, click the *File* menu and click *Run PROFILES.*



Initially, the PROFILES program will display a blank screen with a toolbar showing options to *View Segment Bed Slope* or *Interpolate Segment Slope/Shape*.



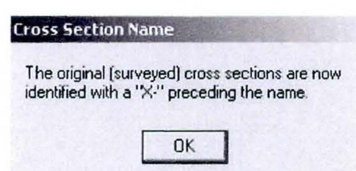
Click the *View Segment Bed Slope* View Segment Bed Slope button to display the initial bed profile. Notice the stair step pattern. This is the bed slope of the channel. It indicates that there are no cross sections between the surveyed cross sections resulting in a flat slope.



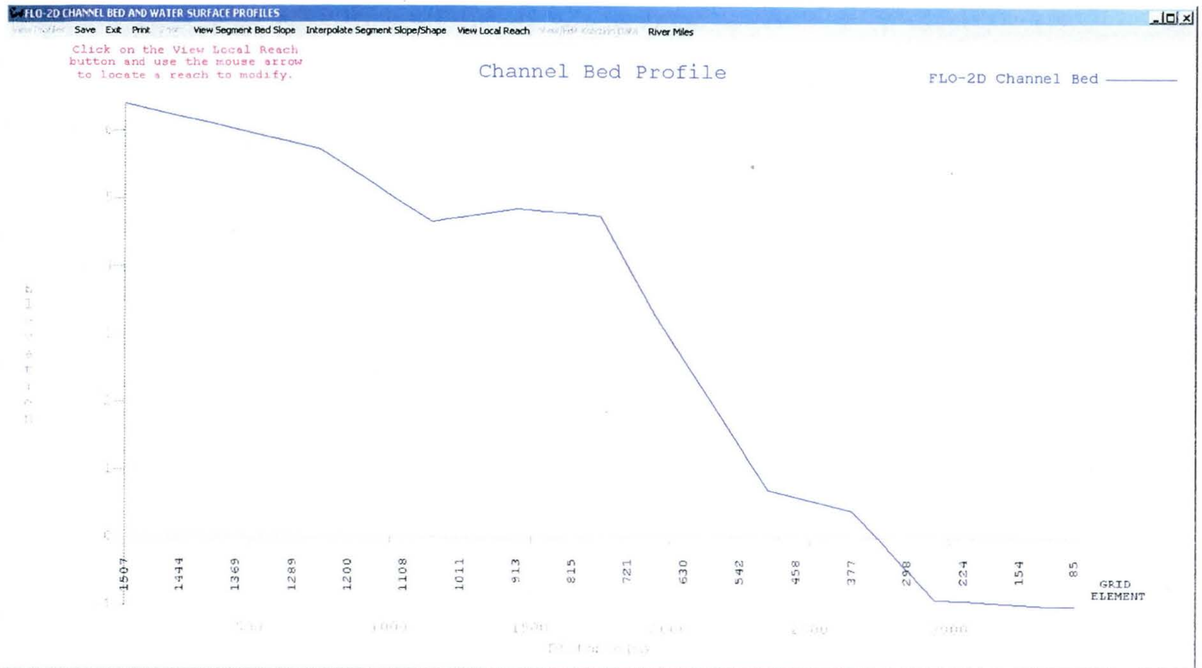
Step 5: Automatically interpolate cross section data

Click the *Interpolate Segment Slope/Shape* 

button and the following dialog box will appear. This states that the original cross section names will begin with an X- to help identify surveyed cross sections. Click *OK*.

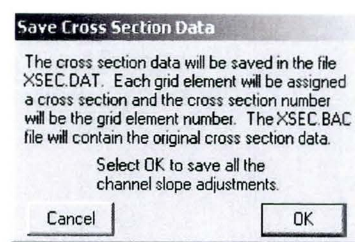


PROFILES will automatically locate the surveyed cross section data and interpolate the cross section geometry and elevation (thalweg slope) to all those channel elements between the surveyed cross sections within the segment. Notice the bed slope of the channel has been smoothed out compared to the previous staircase pattern.

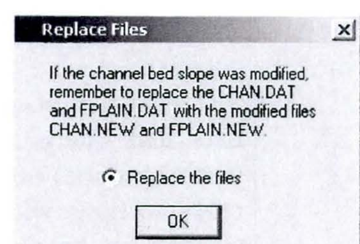


Click the *Save* button and the following dialog box will appear. This box states that the original data file was preserved. Click *OK* and the second box appears. Click the *Replace the files* radio button and click *OK*.

Note: The original cross sections may extend beyond the channel banks across the floodplain. If floodplain cross section stations have elevations beyond the top of bank elevations, those stations should be deleted

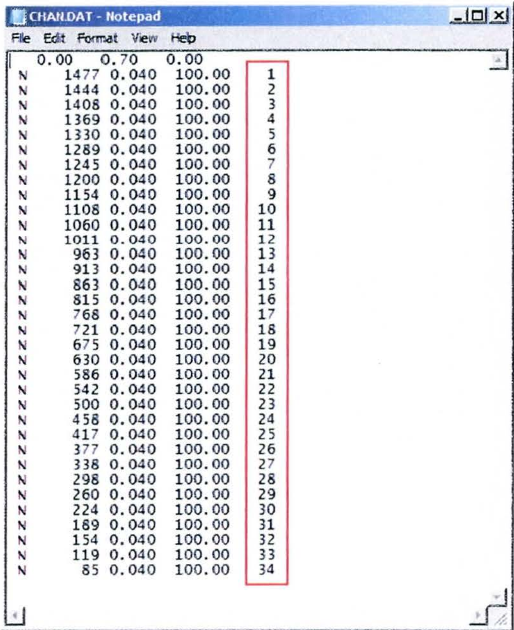


Note: Do not replace the CHAN.DAT and FPLAIN.-DAT files unless backup files have been created.

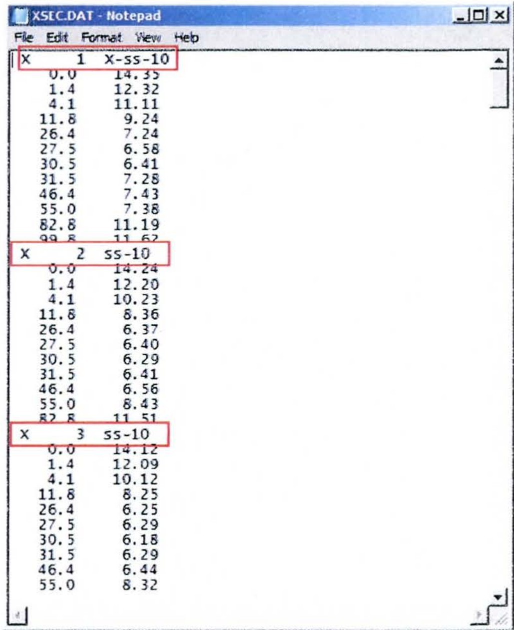


Step 6: Examine the post-interpolation channel data files

Reopen both files in NotePad. Notice that the cross section data is filled in the last column of the CHAN.DAT file and there are now 34 cross sections in the XSEC.DAT file. The cross sections with an X- in the name are surveyed and the others are interpolated.


CHAN.DAT - Notepad
File Edit Format View Help

```
  0.00    0.70    0.00
N     1477 0.040   100.00     1
N     1444 0.040   100.00     2
N     1408 0.040   100.00     3
N     1369 0.040   100.00     4
N     1330 0.040   100.00     5
N     1289 0.040   100.00     6
N     1245 0.040   100.00     7
N     1200 0.040   100.00     8
N     1154 0.040   100.00     9
N     1108 0.040   100.00    10
N     1060 0.040   100.00    11
N     1011 0.040   100.00    12
N      963 0.040   100.00    13
N      913 0.040   100.00    14
N      863 0.040   100.00    15
N      815 0.040   100.00    16
N      768 0.040   100.00    17
N      721 0.040   100.00    18
N      675 0.040   100.00    19
N      630 0.040   100.00    20
N      586 0.040   100.00    21
N      542 0.040   100.00    22
N      500 0.040   100.00    23
N      458 0.040   100.00    24
N      417 0.040   100.00    25
N      377 0.040   100.00    26
N      338 0.040   100.00    27
N      298 0.040   100.00    28
N      260 0.040   100.00    29
N      224 0.040   100.00    30
N      189 0.040   100.00    31
N      154 0.040   100.00    32
N      119 0.040   100.00    33
N       85 0.040   100.00    34
```


XSEC.DAT - Notepad
File Edit Format View Help

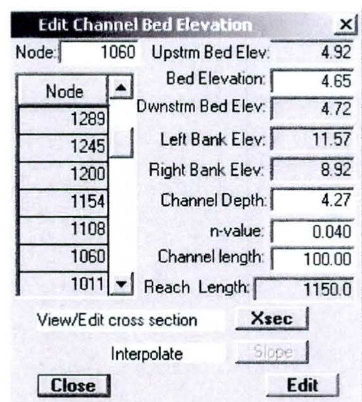
```
X      1  X-ss-10
   0.0     14.35
   1.4     12.32
   4.1     11.11
  11.8      9.24
  26.4      7.24
  27.5      6.58
  30.5      6.41
  31.5      7.28
  46.4      7.43
  55.0      7.38
  82.8     11.19
  99.8     11.62
X      2  ss-10
   0.0     14.24
   1.4     12.20
   4.1     10.23
  11.8      8.36
  26.4      6.37
  27.5      6.40
  30.5      6.29
  31.5      6.41
  46.4      6.56
  55.0      8.43
  82.8     11.51
X      3  ss-10
   0.0     14.12
   1.4     12.09
   4.1     10.12
  11.8      8.25
  26.4      6.25
  27.5      6.29
  30.5      6.18
  31.5      6.29
  46.4      6.44
  55.0      8.32
```

Step 7: Edit cross section data

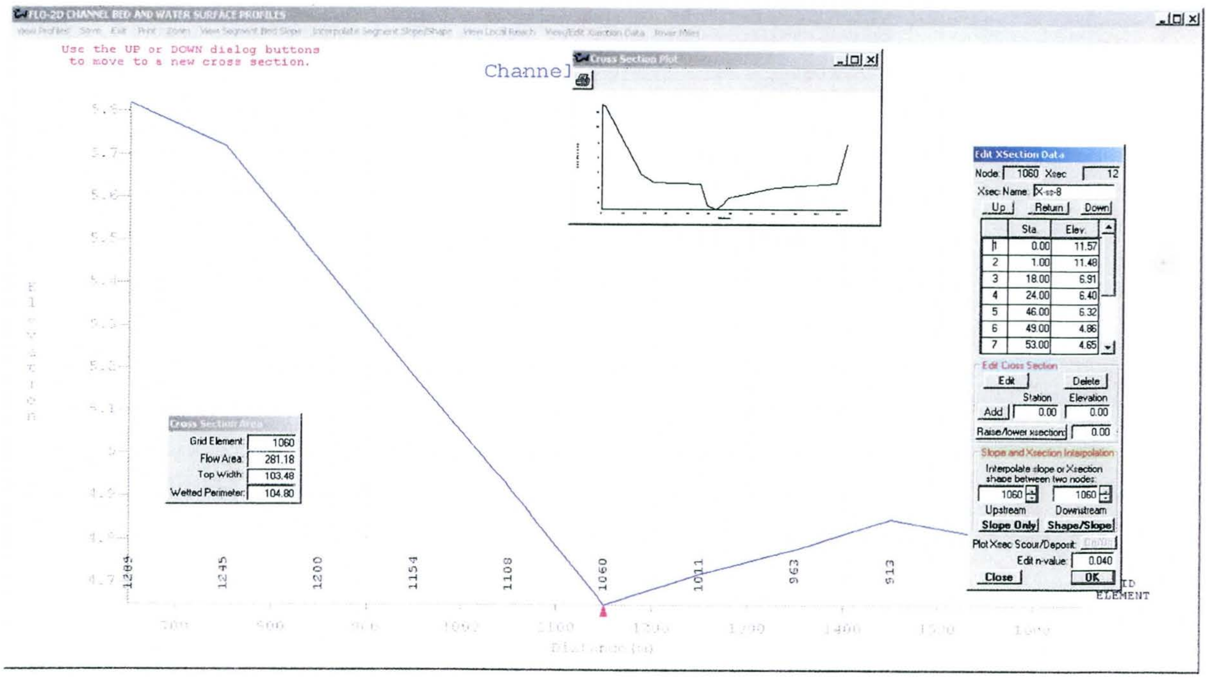
Click on the *Pre-Processor* pull down menu and click on *PROFILES*.

On the PROFILES window, click *View Segment Bedslope* button and then click the *View Local Reach* [View Local Reach] button on the *Toolbar*. Notice the cursor changes from a standard arrow to a thin vertical arrow. Click the location shown on the screen to zoom in on a reach of 10 channel elements (5 upstream and 5 downstream of the cursor location).

Click on *View/Edit Xsection Data* View/Edit Xsection Data button to view the following dialog box.



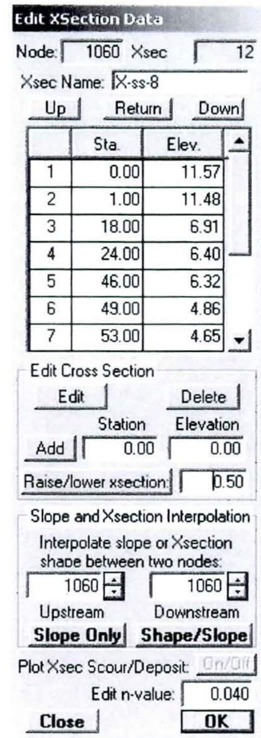
Next, click on the *Xsec* Xsec button in the dialog box to view the cross section data and plot.



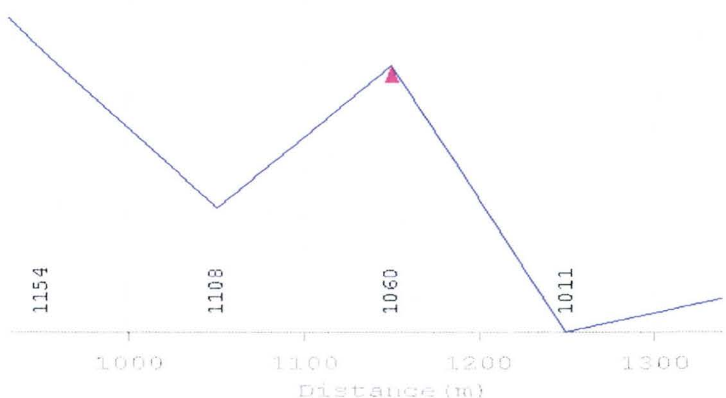
At this point review some of the other cross sections by clicking on the "Up" and "Down" buttons in the dialog.

Use the list to edit the individual cross section station data or add and delete stations. As an example of editing the cross section, we are going to raise the elevation of this cross section and then manually interpolate between it and its neighboring cross sections.

Enter a *0.5* in the text box next to the *Raise/Lower xsection* Raise/lower xsection button and click the button. One half foot has just been added to each station elevation point in this particular cross section.



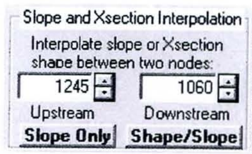
Now see that there is a spike in channel bed slope.



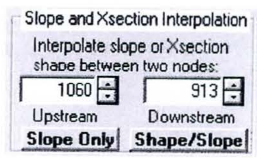
Step 8: Manually interpolate between two cross sections

To adjust the slope and shape for the revised cross section, it is necessary to interpolate between the cross section that was modified and its upstream and downsstream neighbors.

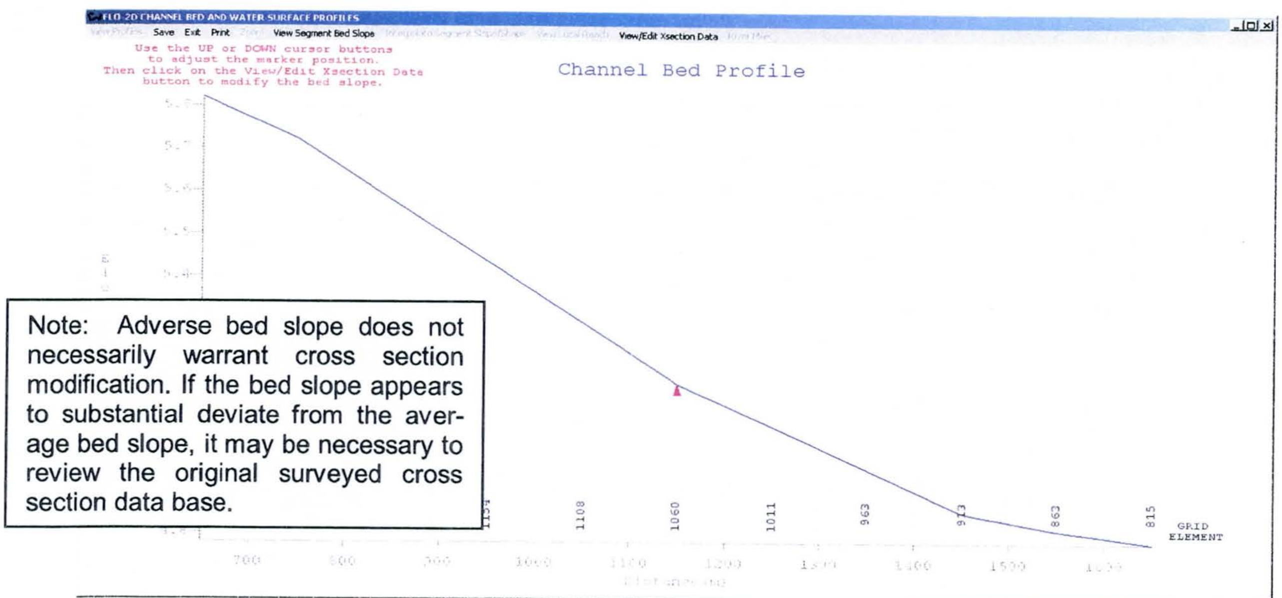
In this case, notice that the changes were made to a surveyed cross section *X-ss-8 (Node 1060)*. Click the *Up* button *4* times to reach cross section X-ss-9. Now enter *1060* in the right interpolation box and click the *Shape/Slope* button.



Now click the *Down* button *7* times to cross section *X-xx-7*. Enter a *1060* in left interpolation box and click the *Shape/Slope* button. To retain the cross section shape and only edit the bed slope, click the *Slope Only button.*



Click *Close* on the *Edit Xsection Data* dialog box and click *Close* on the *Edit Bed Elevation* dialog box. Notice the bed slope no longer dips at channel element *1060*. Once editing is completed save and exit the PROFILES program.



Note: Adverse bed slope does not necessarily warrant cross section modification. If the bed slope appears to substantial deviate from the average bed slope, it may be necessary to review the original surveyed cross section data base.

Summary

The PROFILES lesson was designed to show how to edit cross section and bed slope data. This program is a crucial tool when developing a natural channel model. PROFILES can be used to interpolate the channel data between cross sections and to edit the station elevation cross section data.

Review Questions

1. Rectangular or trapezoidal cross section can be assigned with the natural shape cross section option? (True or False) ____________
2. When it is necessary to interpolate the channel cross sections?

   ____

   ____

   ____

   ____

3. If a cross section has a levee that is higher than the channel top of bank, how far does the cross section extend?

   ____

   ____

4. Channel flood routing primarily depends on which one of the following?
   - ☐ Channel length inside the grid element
   - ☐ Channel cross section flow area
   - ☐ Channel n-value
   - ☐ Channel energy slope
   - ☐ Channel surface area
5. The channel geometry is measured to the lowest top of bank? (True or False)

   ____

6. The last channel element in a segment must be a channel outflow node? (True or False)

   ____

7. The automated cross section interpolation when building the channel data files only affects the channel geometry? (True or False) ____________

LESSON 6 – CREATING FLOOD MAPS WITH MAPPER

Part 1. General Mapping

Flood Maps, Layers and Exclusion Polygons

Overview

The MAPPER program generates numerous high resolution flood routing plots ranging from maximum flow depths and velocities to damage assessment. In this lesson, use the MAPPER program to graphically display model results, generate flood mapping, and modify layer attributes.

Required Data

For this lesson, the FLO-2D data files *.DAT and output files *.OUT found in the Lesson 6 subdirectory and the elevation data *.PTS in the Lesson 1 subdirectory are needed.

File	Content	Location*
Fplain.dat	Digital Terrain Model file	*Example Projects\Lesson 6*
dtm.pts	Digital elevation points file	*Example Projects\Lesson 1*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

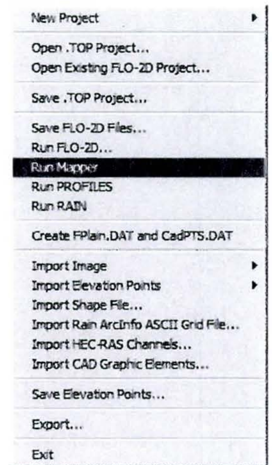
Analyzing output data and creating flood maps with MAPPER will require the following steps.

- Open the MAPPER program and load project;
- Display the grid system;
- Plot maximum flow depth data;
- Plot and customize shaded contours;
- Modify layer attributes and legend properties;

- Add aerial images;
- Create DTM shaded contour layer and export GIS data;
- Create an exclusion polygon.

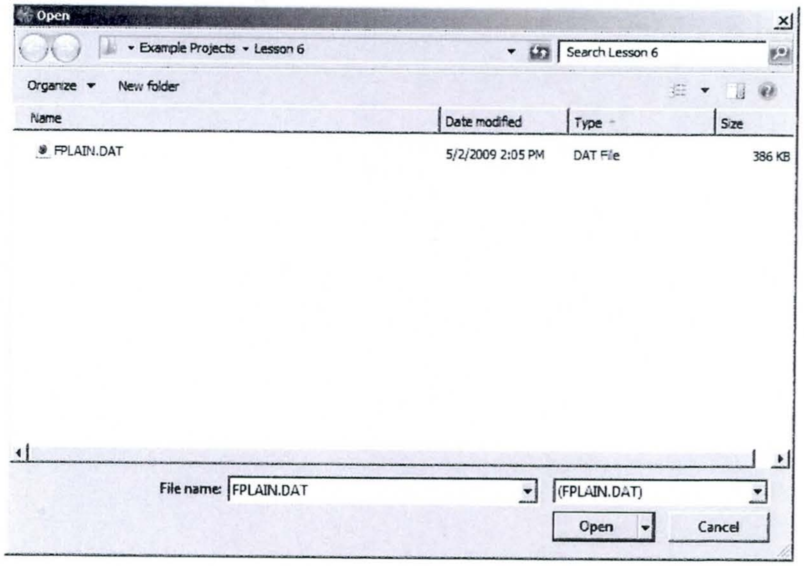
Step 1: Open MAPPER and load the Lesson 6 project

In the GDS, select the *File* menu from the toolbar and click *Run MAPPER.*

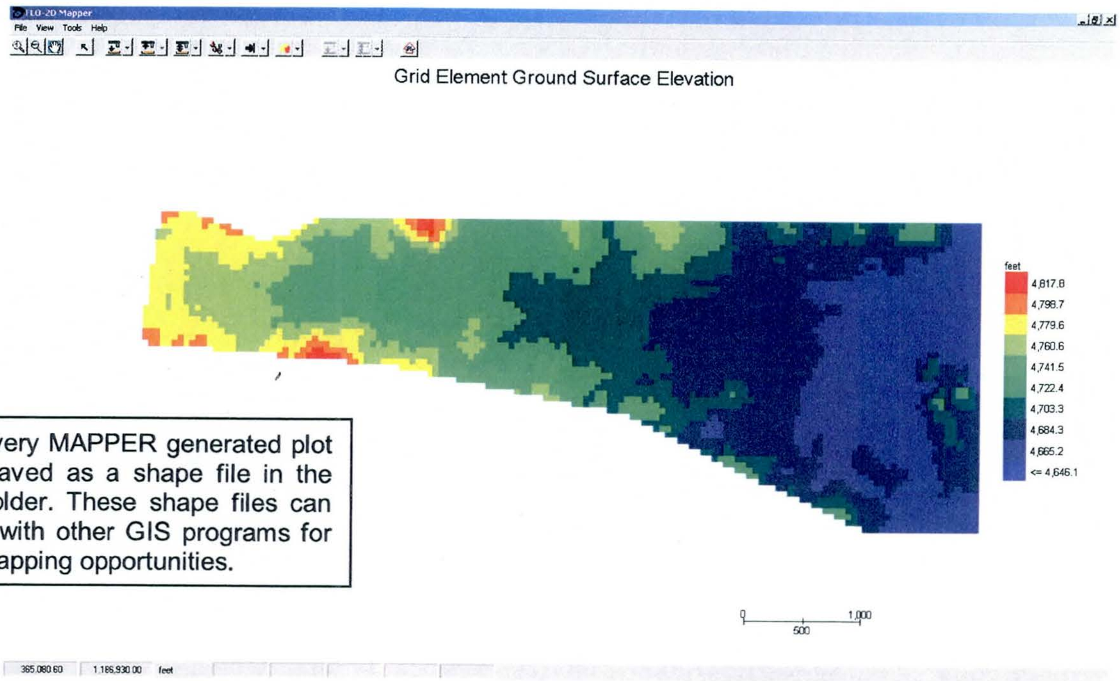


On the Mapper Toolbar, open the project output files by clicking on *File* and *Read FLO-2D Results…* as shown below.

Locate the project file folder and click on the *Open* button.



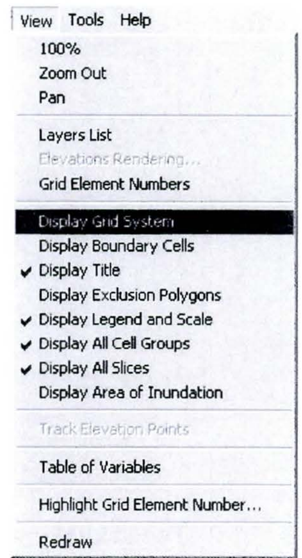
The grid element topography will be displayed.

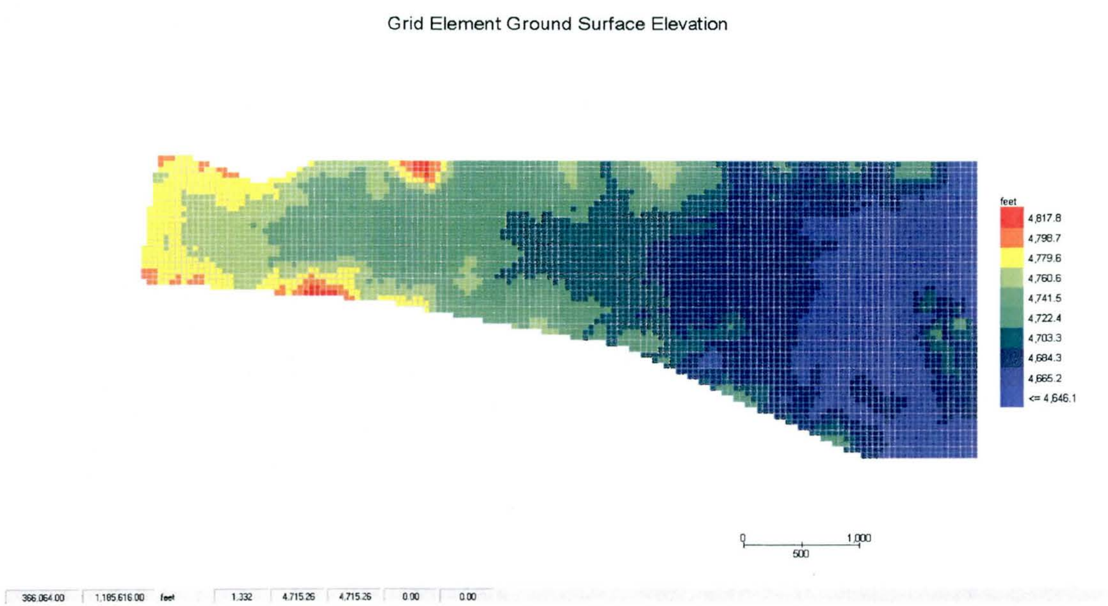


Note: Every MAPPER generated plot is also saved as a shape file in the project folder. These shape files can be used with other GIS programs for further mapping opportunities.

Step 2: Display the grid system

The grid system can be displayed with all MAPPER graphics. Click *View/Display Grid System*.

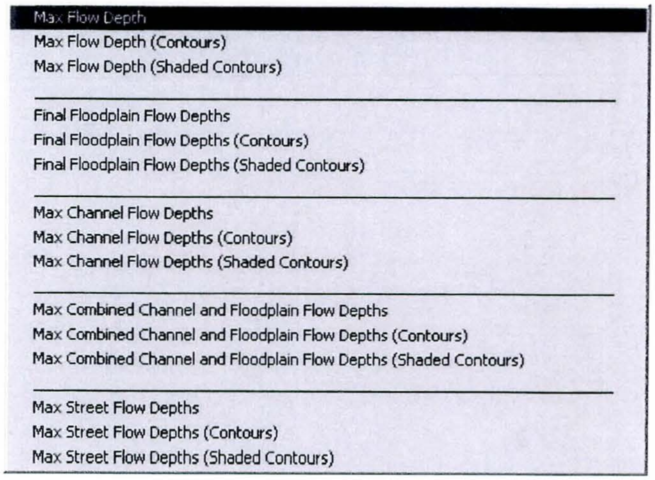




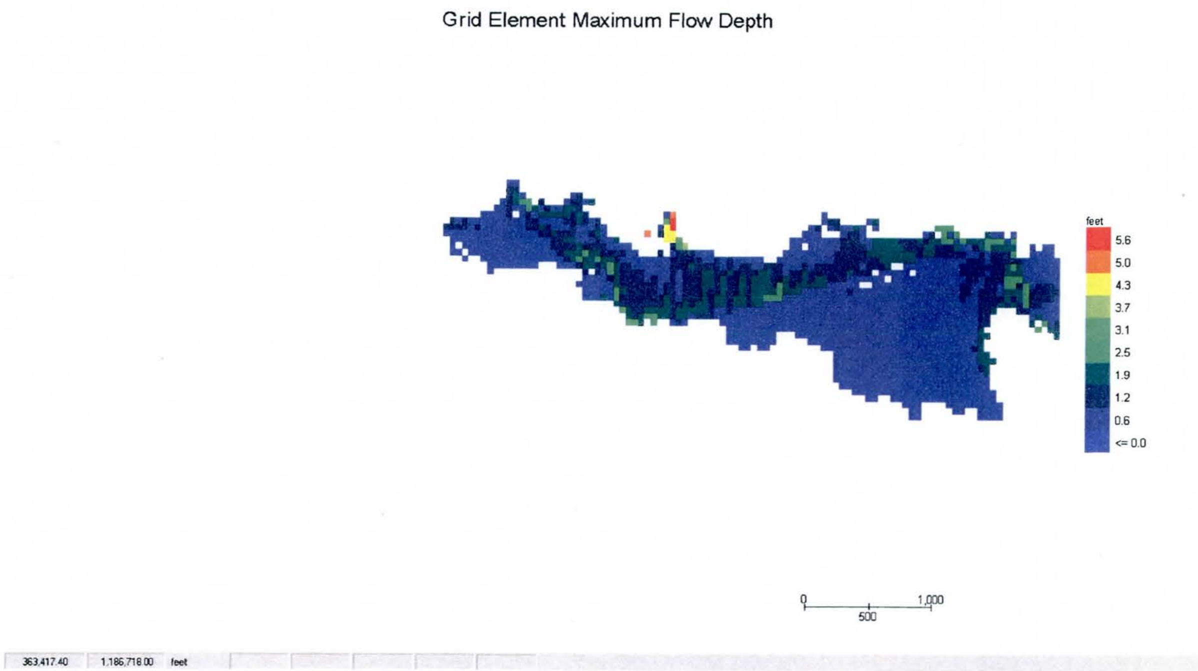
Turn off the grid system using the same procedure.

Step 3: Plot maximum flow depth at cell

It is necessary to plot the grid element maximum flow depth to initiate other mapping options such as animation. Click on the icon *Max Flow Depth at Cell* pull down menu and clicking *Max Flow Depth*.



The grid element maximum flow depths are displayed.

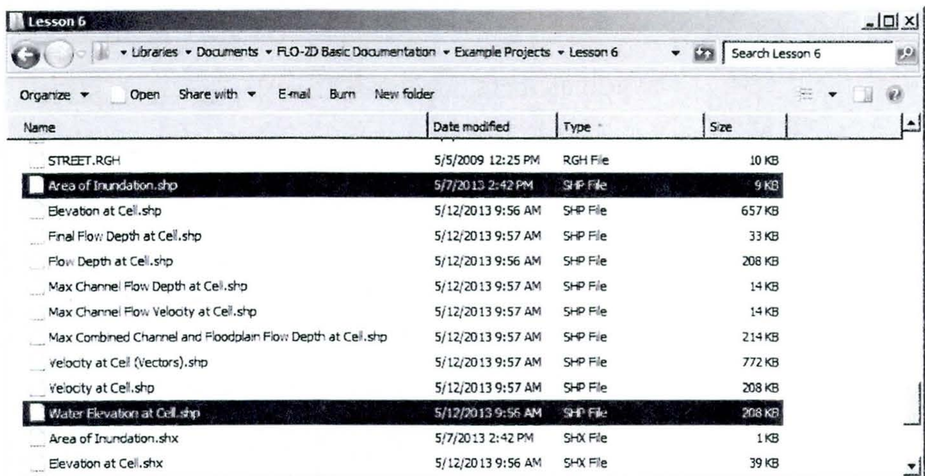


Plot the many grid element maps using the icon tool bar at the top of the screen. This includes.

- Ground surface elevation
- Maximum water surface elevation
- Final floodplain depths
- Maximum channel flow depths
- Maximum combined channel and floodplain depths
- Maximum street depths

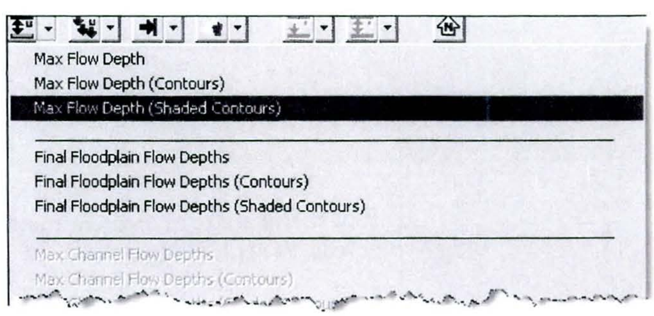
- Maximum floodplain velocity
- Maximum street velocity
- Maximum channel velocity
- Final floodplain velocity
- Duration of inundation
- Maximum impact force
- Maximum static pressure
- Maximum specific energy
- Levee freeboard deficiency

The plots are saved as shape files as soon as they are loaded and are found in the project folder.

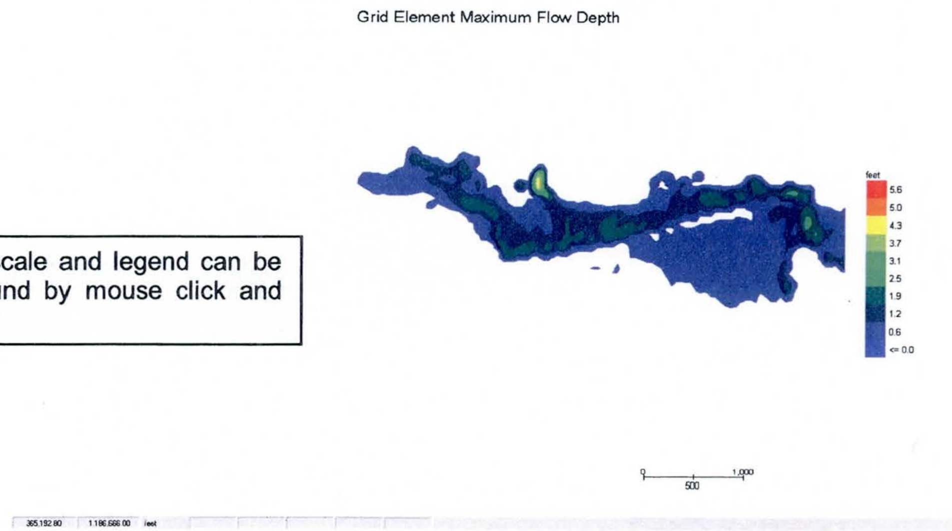


Step 4: Plot and customize max flow depth (shaded contours)

The maximum grid element flow depths can be plotted as shaded contours.



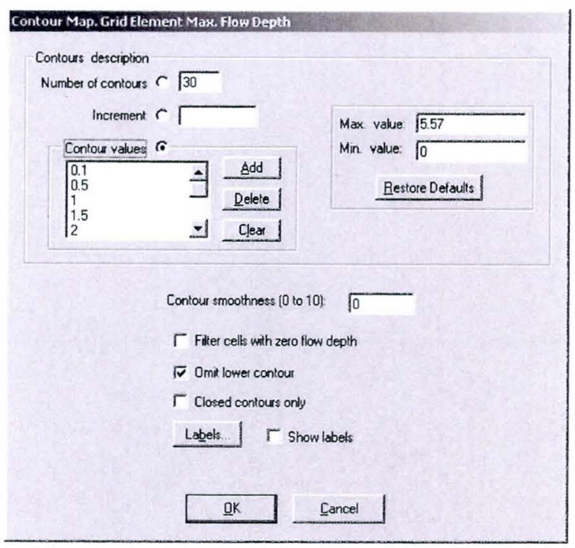
The flow depths are the maximum depths that were computed regardless of the time of occurrence. The maximum depth plots therefore display the maximum area of inundation.



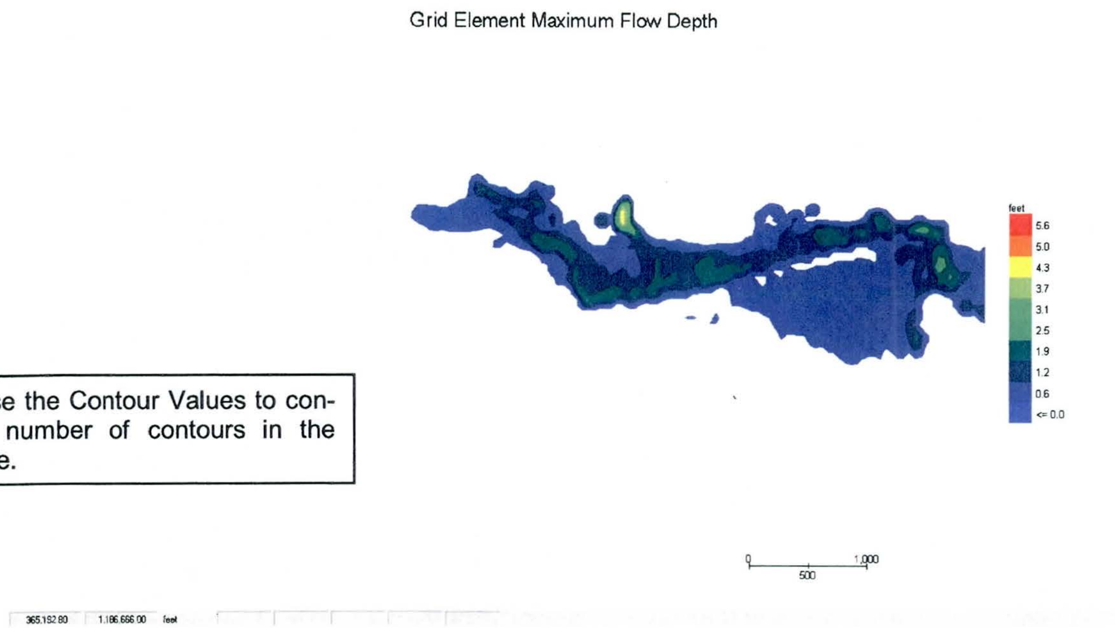
Note: The scale and legend can be moved around by mouse click and drag.

The shaded contour plot can now be customized by selecting more contours, identifying a prescribed contour interval or by specifying the individual contour intervals as shown in the following dialog box.

To display this dialog box, right click anywhere on the contour plot. In this instance, the *Add* button was selected and contour intervals every *0.5* ft were added to the list of contour values along with an initial contour of *0.1* ft.



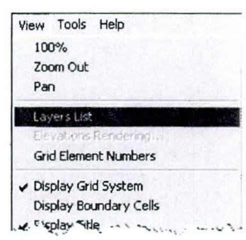
Click *OK* to display the shaded contour. The area of inundation is smaller because the lower contour was omitted.



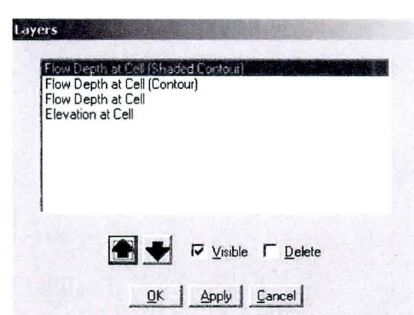
Note: Use the Contour Values to control the number of contours in the shape file.

Step 5: Modify layer attributes and legend properties

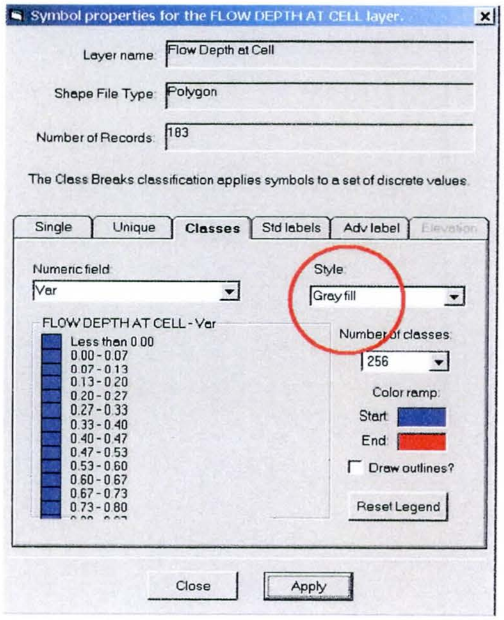
To enhance the shaded contour plots create transparent shaded contours. To make the shaded contours transparent, open the layer's list. Click *View/Layers List*.



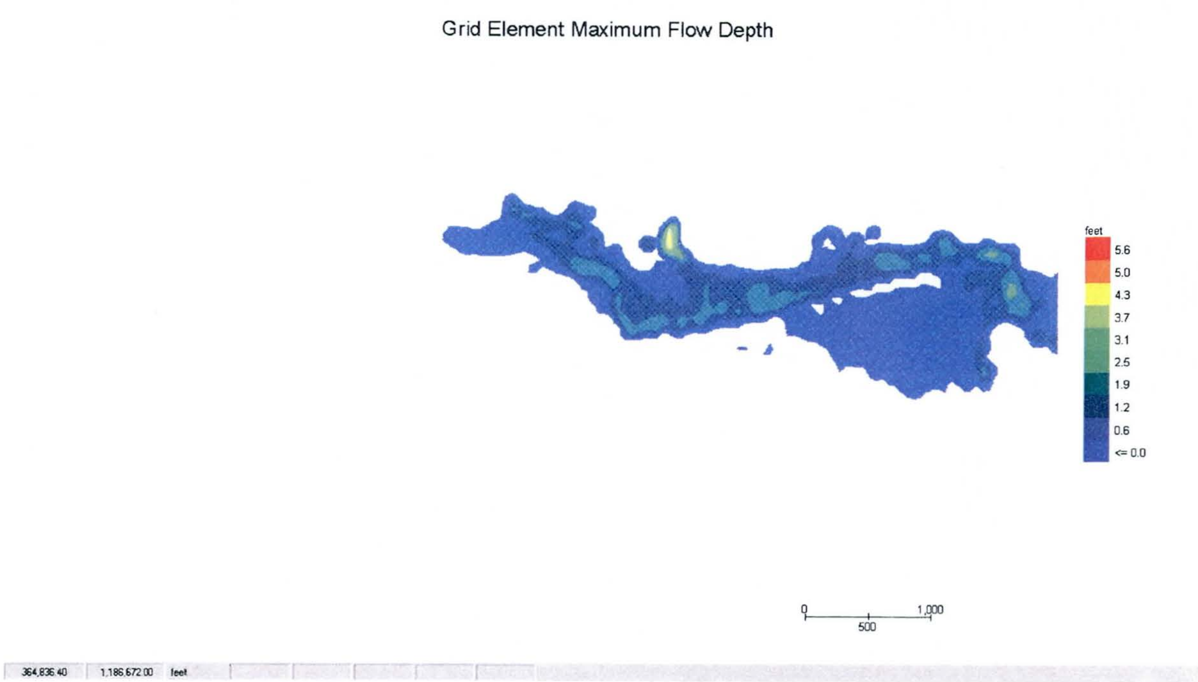
Next double click on the *Flow Depth at Cell (Shaded Contour)* layer as shown below.



Select the *Gray fill* or *Light gray fill* in the *Style* list of *Symbol Properties* dialog box. To accept changes, click *Apply* and *Close*.



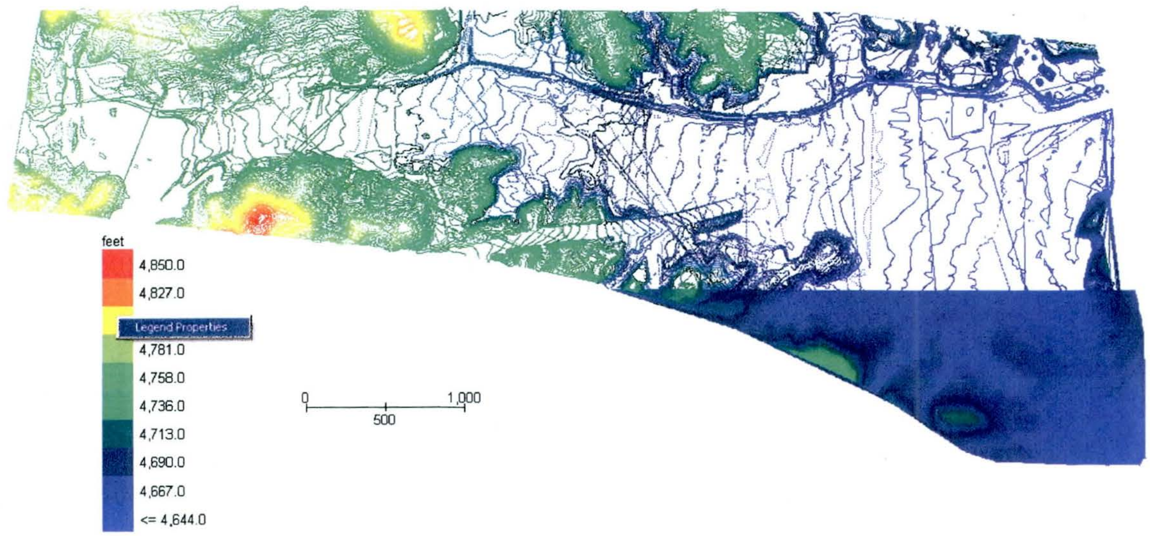
Also close the *Layer List* dialog box and the following shaded contour plot is generated.



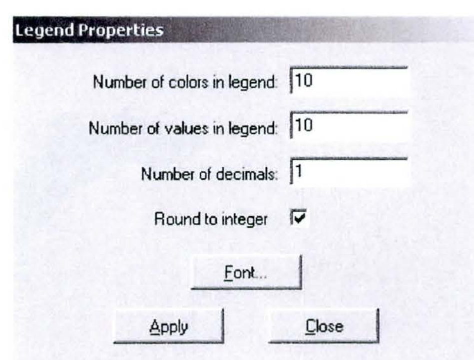
The legend and scale properties and location can be adjusted. This function works well if while capturing a screen shot of the MAPPER window.

Click and drag the *legend* and *scale* to a new location on the window. Right click the *legend* and click *Legend Properties*.

DTM Points Ground Surface Elevation

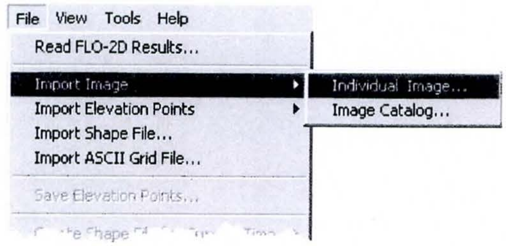


Change the number of decimal places, color and size of the font as well as other legend properties with this dialog box.

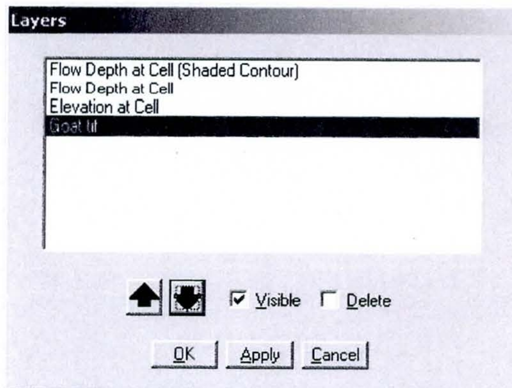


Step 6: Add aerial image to background

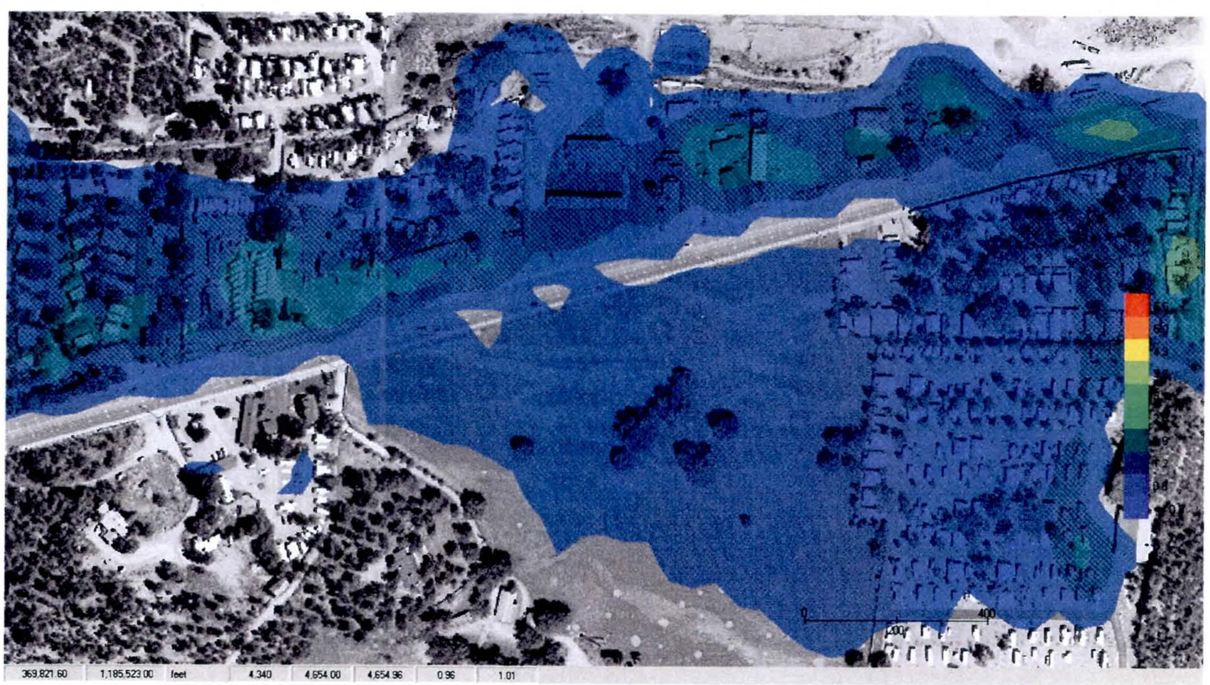
Import the aerial background images. Click *File/Import Image/Import Individual Image* to load the *Goat.tif* image in the *Lesson 1* Folder.



Click *View/Layers List* and highlight the *Goat* image. Move it to the bottom of the list using the arrow  button.



Click *Apply* and *OK*.

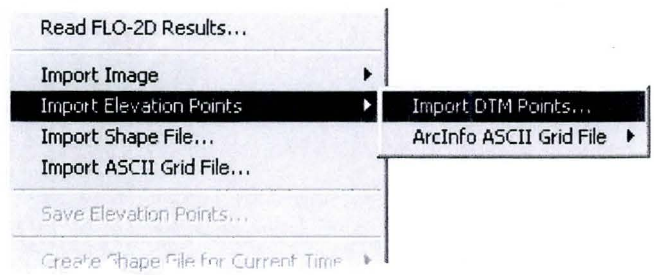


Step 7: Create depth to DTM point plot

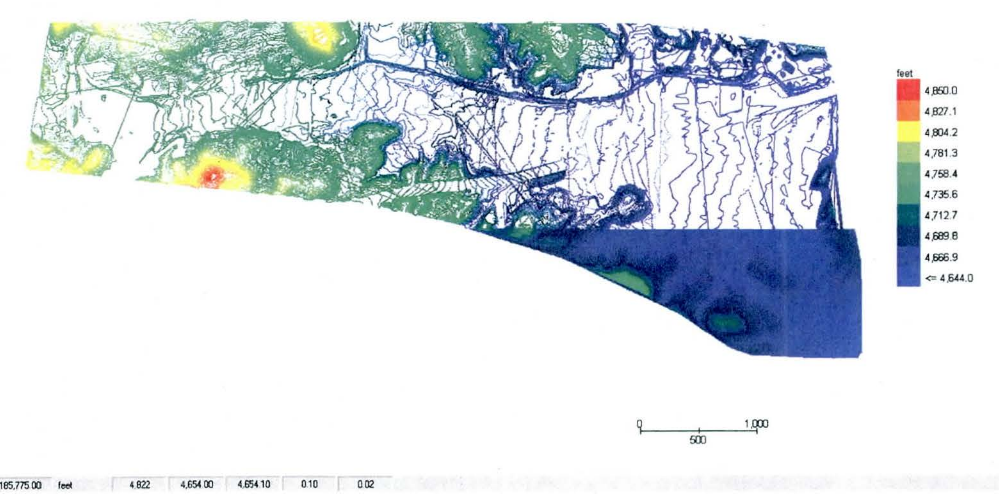
Enhance the resolution of flood hazard maps by plotting shaded contours of the DTM point flow depths. First the DTM points will have to be imported. Then MAPPER will subtract the DTM ground elevation points from the grid element predicted water surface elevations. Finally, the shaded contour plots can be created.

Click *Import Elevation Points/Import DTM Points…* in the *File* menu. The *dtm.pts* file is located in the *Lesson 1* folder.

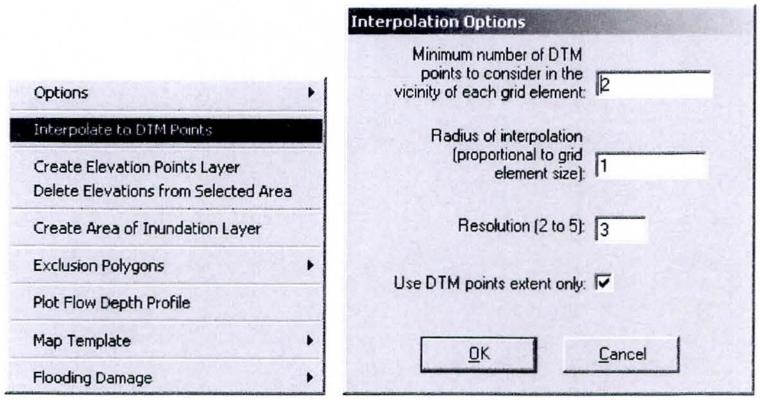
Warning: This step can be time consuming.

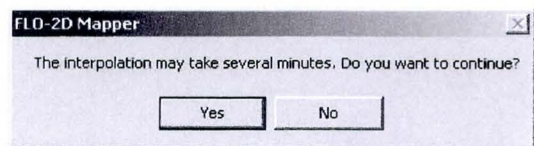


DTM Points Ground Surface Elevation



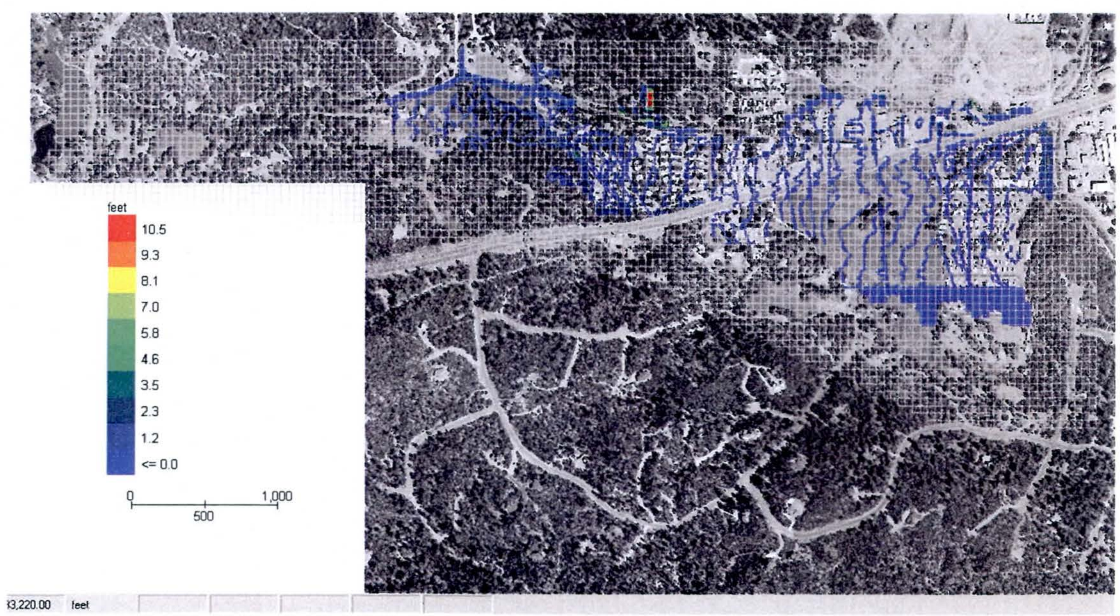
Now interpolate the flow depth to the DTM points and subtract the DTM point ground elevation from the grid element predicted maximum water surface elevation. Click *Tools/Interpolate to DTM Points*.



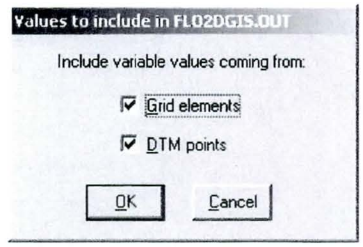


Click *OK* and *Yes* and wait for the interpolation process to complete.

To plot a shaded contour representation of the maximum interpolated flow depth over the DTM points, click the *DTM Points Max Flow Depth* button. Note that some of the DTM points were based on topographic contour lines.



The DTM point maximum flow depths can be saved to file by clicking on the *FLO2DGIS* command in the *File* pull down menu. The *FLO2DGIS.OUT* file will contain the grid element number, the x- and y-coordinates and the maximum flow depth over the DTM point. This file can be imported to ARCGIS to utilize additional mapping tools.



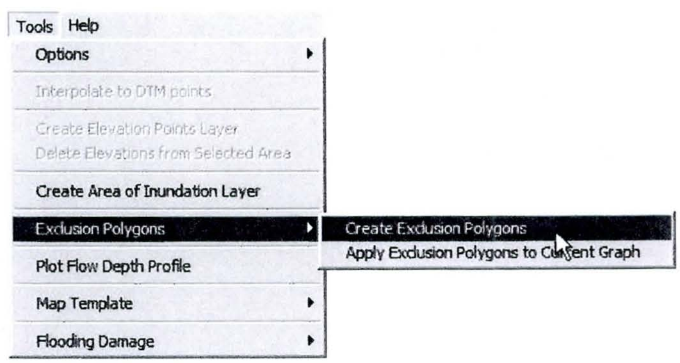
Click *OK* and *Save* to create the *FLO2DGIS.OUT* file.

Step 8: Create an exclusion polygon

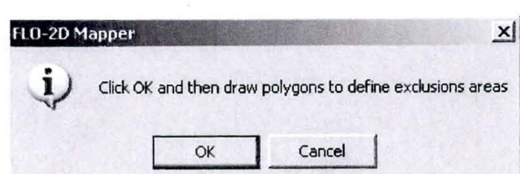
Exclusion polygons can be used to edit shaded contour plots. Constructing shaded contours can result in the mapped area exceeding physical features on the map. This occurs when the contour plots are not controlled by break-lines or by the contour interval. To improve the appearance of the shaded contours, delete a portion of a contour shapes. An example where the use of exclusion polygons is appropriate is when a shaded contour spills over a levee, bluff or raised highway.

Load the *Max Flow Depth at Cell (Shaded Contour)* layer by clicking *Max Flow Depth at Cell* pull down menu and clicking *Max Flow Depth (Shaded Contour)*.

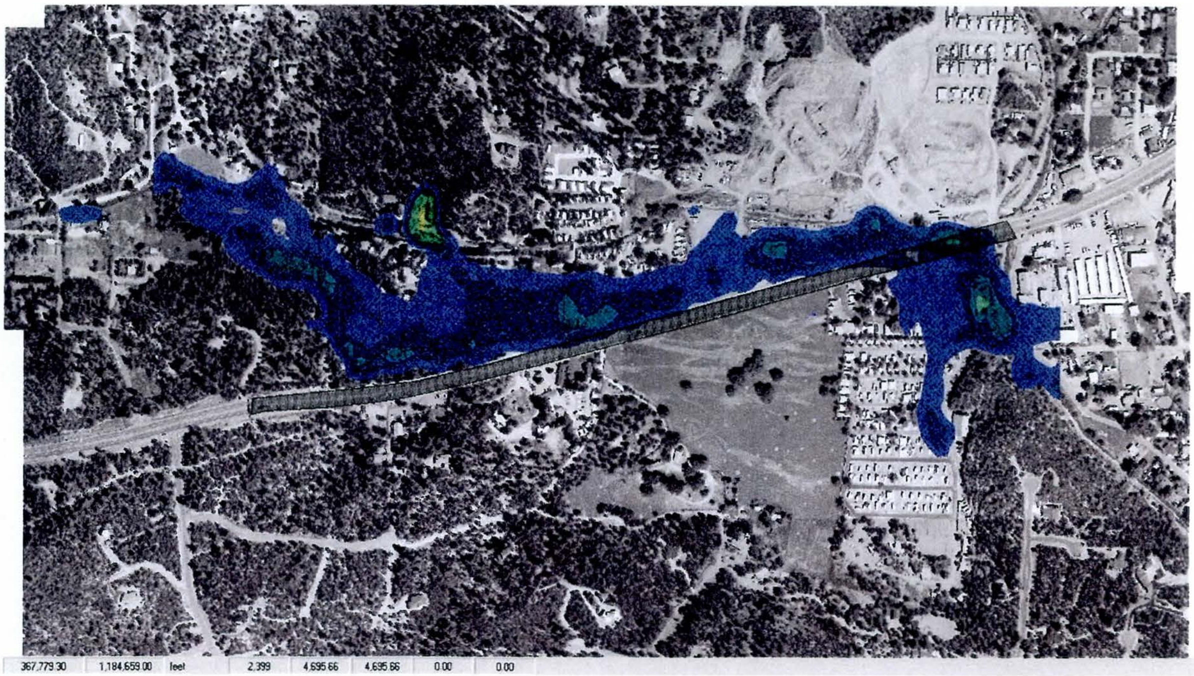
Click the *Exclusion Polygons/Create Exclusion Polygons* command on the *Tools* menu.



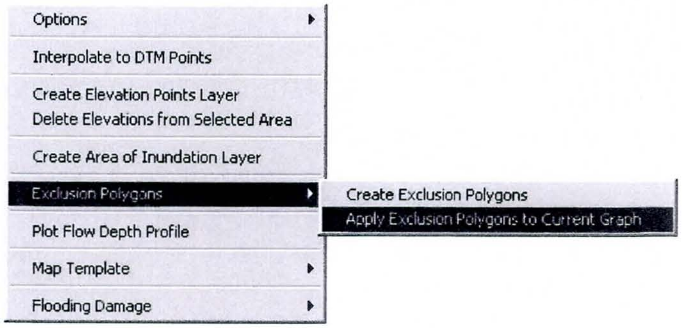
Click *OK* and draw one or more polygons on the flow depth shaded contour plot.



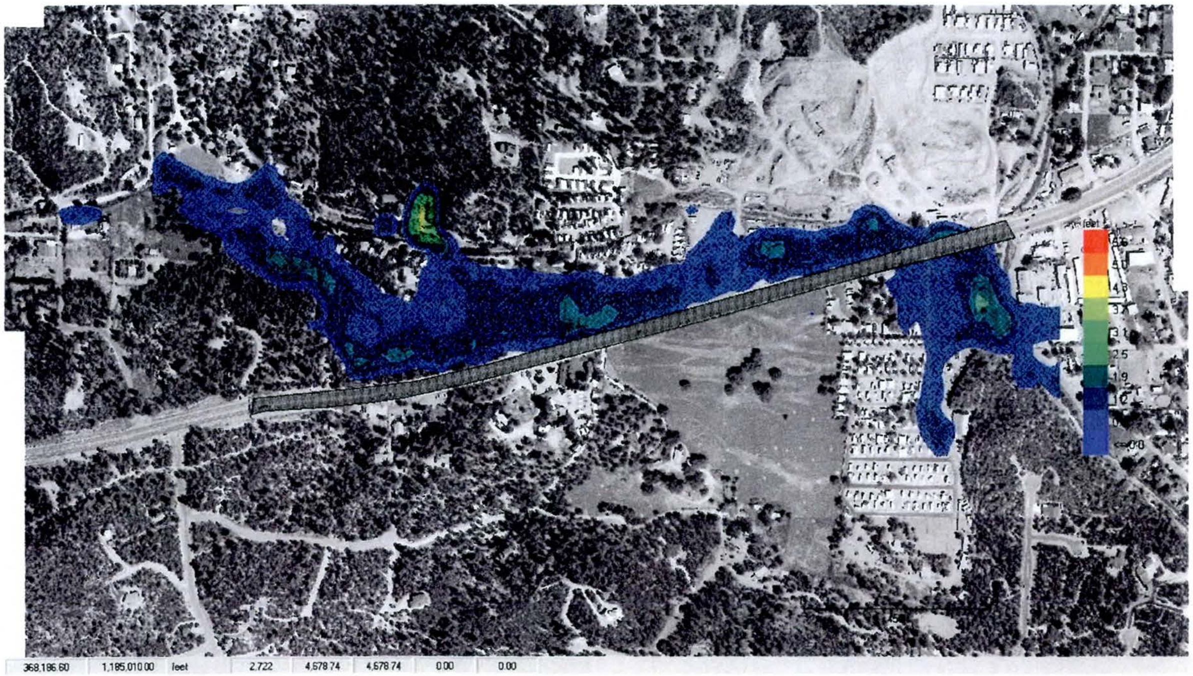
Assume for this step that the highway is raised and that water cannot overtop it. In the following image, all of the flow depth that coincides with the highway is a contour spill. Remove the contour spill by deleting the portion of the plot that coincides with the highway. The exclusion polygons define the area that is erased from the shaded contour plot. The image below shows a polygon that will edit the highway portion of the shaded contour.



Click *Tools/Exclusion Polygons/Apply Exclusion Polygons to Current Graph*.

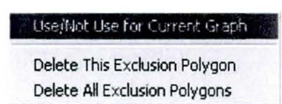


Click *View/Display Exclusion Polygon*. The portion of the shade contour within the polygon was deleted.

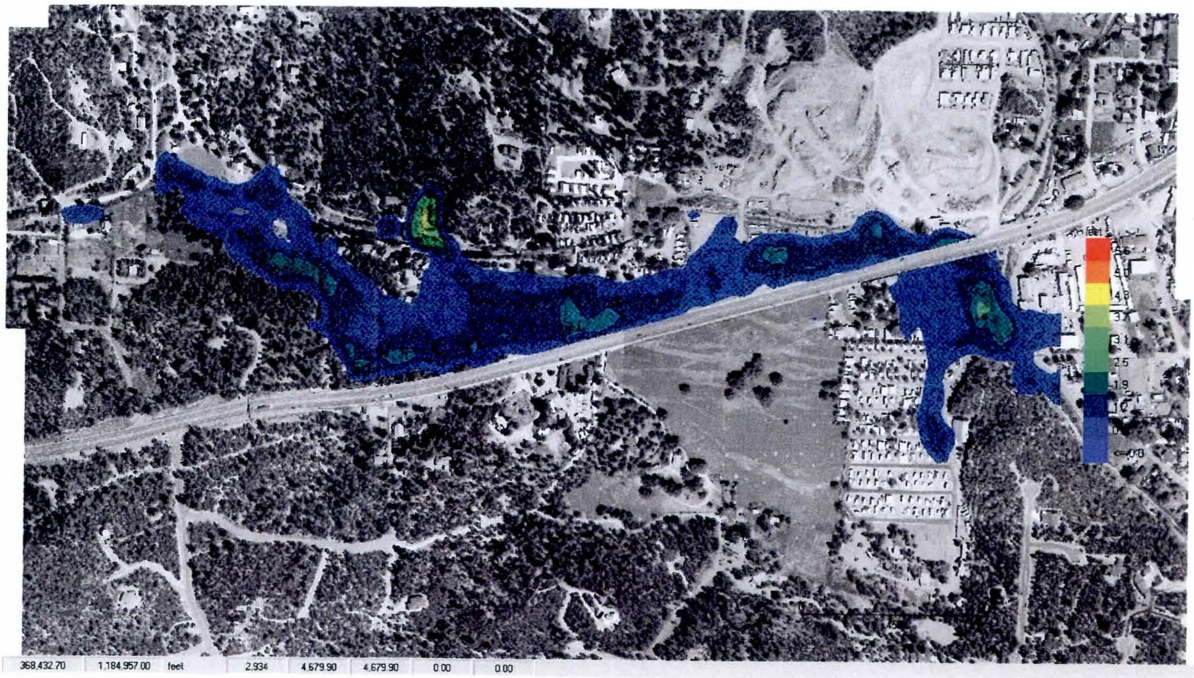


The exclusion polygons can be used multiple times. Just load a shaded contour and click *Tools/Exclusion Polygons/Apply Exclusion Polygons to Current Graph*.

To control the polygon click its border. Click the *Use/Not Use for Current Graph* to select or deselect the polygon that was clicked. It's also possible to delete the polygons with this tool.



The following image shows the shaded contour with the highway portion removed. The *Flow Depth at Cell (Shaded Contour)* shape file was automatically saved and will also reflect these changes.



To revert back to the original shaded contour plot, close and reopen MAPPER and start fresh.

Part 2. Advanced FLO-2D Results Analysis and Mapping

2a. Flood Animation

Overview

The MAPPER program can be used to create the model flood animations. This lesson will show how to view the FLO-2D simulation as an animation of the floodwave progression.

Required Data

Use the FLO-2D data files *.DAT and output files *.OUT including the animation *.OUT files found in the *Lesson 6* subdirectory.

File	Content	Location*
FPLAIN.DAT	Digital Terrain Model file	*\Example Projects\Lesson 6*
TIMEDEP.OUT	Animation file	*\Example Projects\Lesson 6*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

The following steps are necessary to run the flood animation in MAPPER and view temporal results.

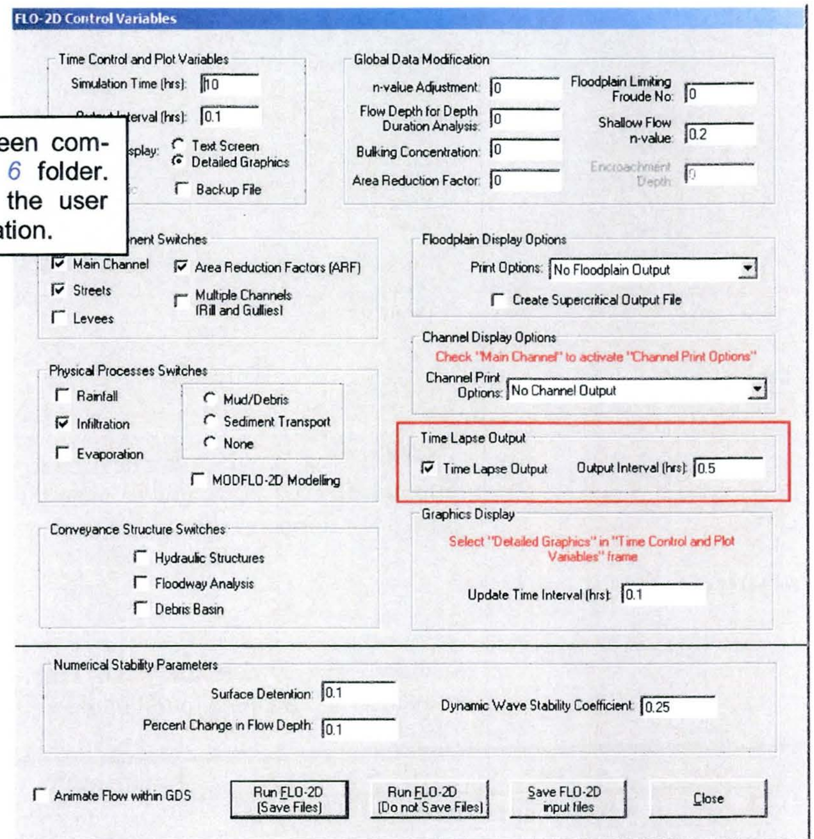
- Set the animations controls in the CONT.DAT prior to the flood simulation.
- Open MAPPER and load *Lesson 6*;
- Run a flood animation;
- Run a velocity animation;
- Create time plots;

Step 1: Set Animation Controls GDS - Run FLO-2D Simulation

Set *ITIMDEP = 1* and TIMDEP = 0.5 in GDS Control Panel. This generates the temporal output files during the FLO-2D simulation

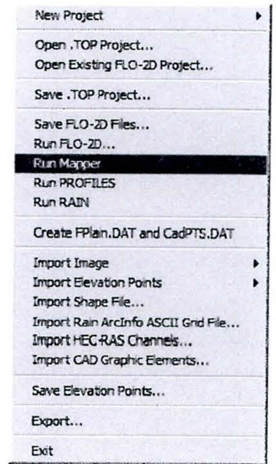
Run FLO-2D to completion.

NOTE: Step 1 has been completed in the *Lesson 6* folder. This step is to show the user how to set up the animation.

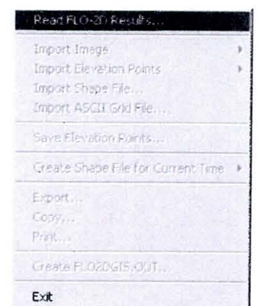


Step 2: Open MAPPER and load the Lesson 6 project

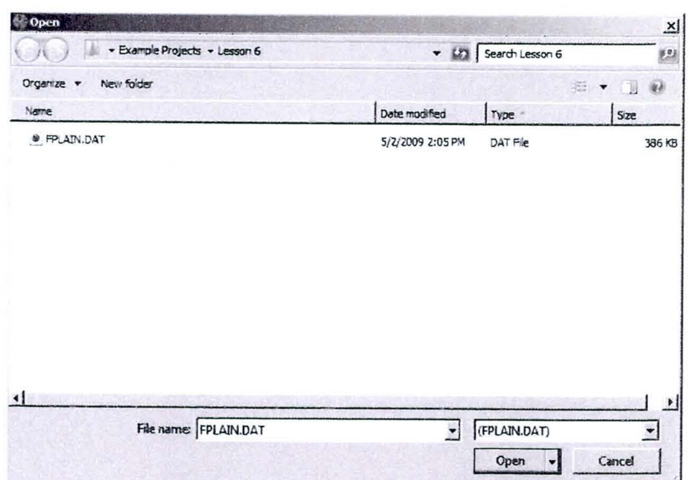
In the GDS, select the *File* menu from the toolbar and click *Run MAPPER.*



In Mapper Open the project output files by clicking on *File* and *Read FLO-2D Results...* as shown below.

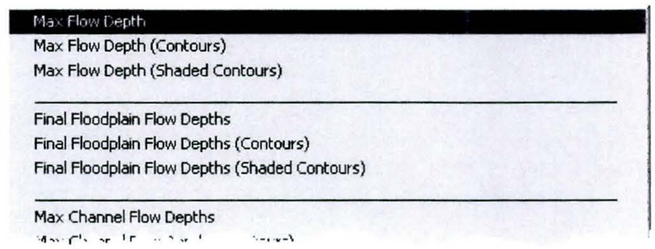


Locate the project file folder and click on the *Open* button.



Step 3: Run a flood animation

The FLO-2D simulation can viewed as an animation of the floodwave progression using this tool. First it is necessary to plot the *Max Flow Depth*.

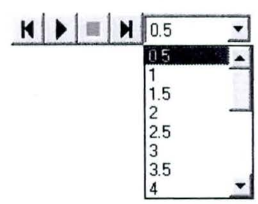


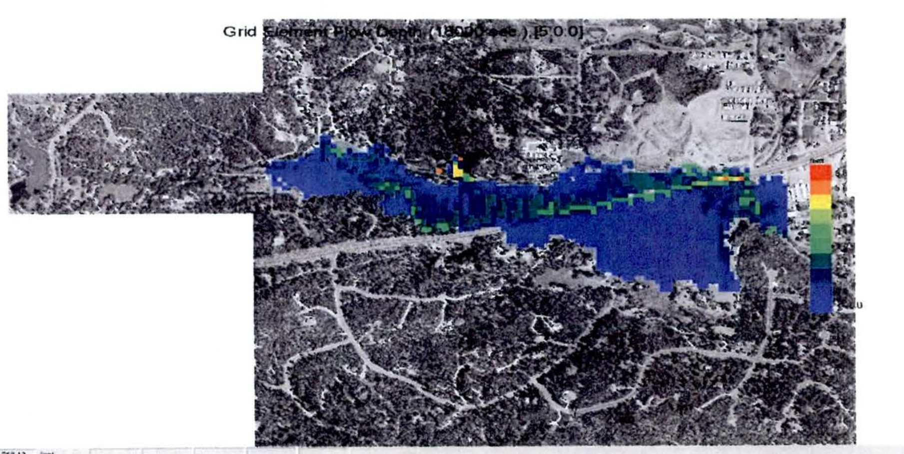
The maximum flow depth for the grid element plot activates the flood animation controls on the tool bar.

The animation controls from left to right are.

Previous View *Play* *Stop* *Next View*

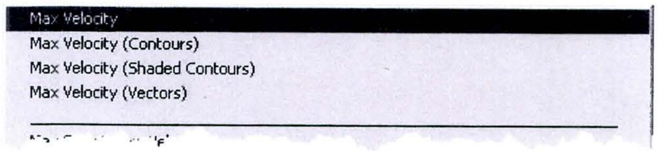
The *Previous View* and *Next View* buttons can be used to manually control the speed of the animation. Click on the *Play* button to begin the flood animation. Click on *Stop* then *Previous View* or *Next View* to view a specific time interval of the flood simulation. The *drop down* list on the right enables the user to select a specific time step to display in the flood simulation.



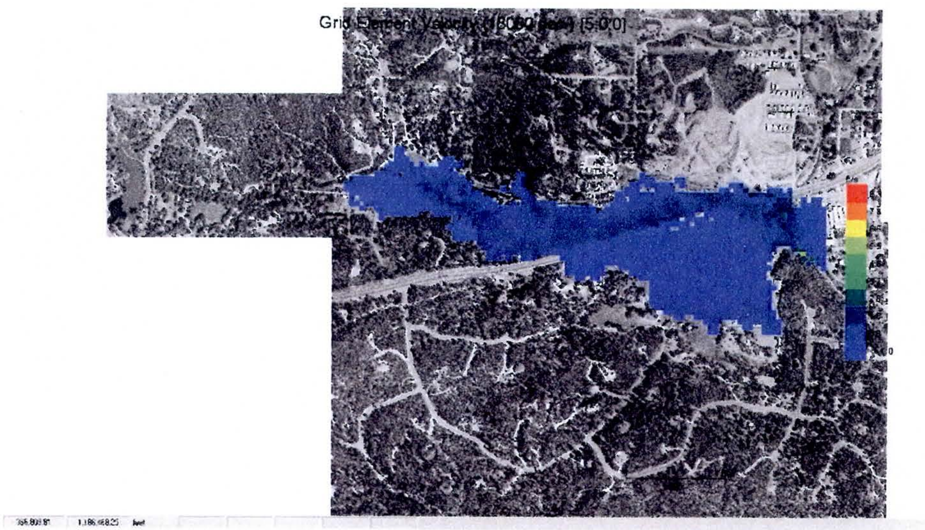


Step 4: Run a velocity animation

The flow velocity animation can also be run in the same sequence by clicking on the Grid Element Max Velocity and starting the animation with the Play button.

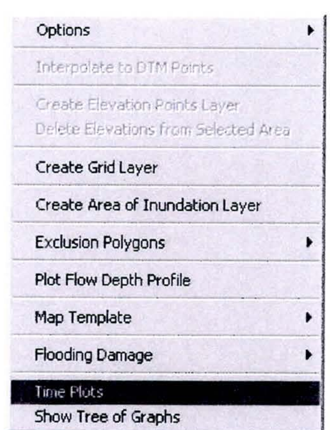


Follow the same procedure for running the flow depth animation.

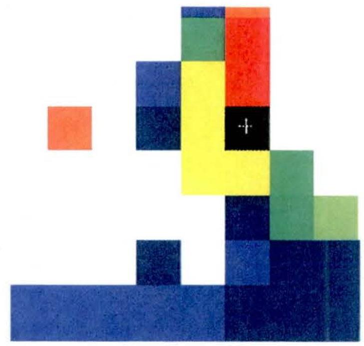


Step 5: Create time plots

Following the animation, it is possible to view a plot of the flow depth versus time or velocity versus time (output interval) for a specific grid element, select *Tools/Time Plots*.

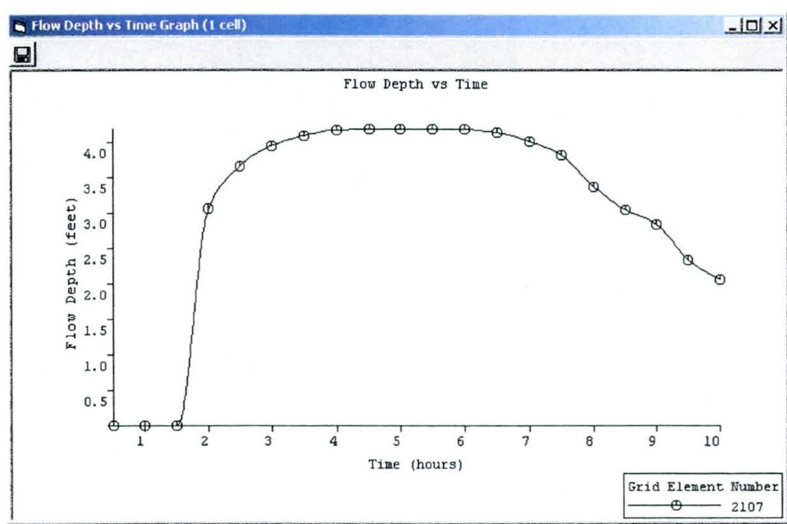


A cursor cross (+) is displayed that can be used to select individual grid elements as shown below. Multiple elements can be selected.

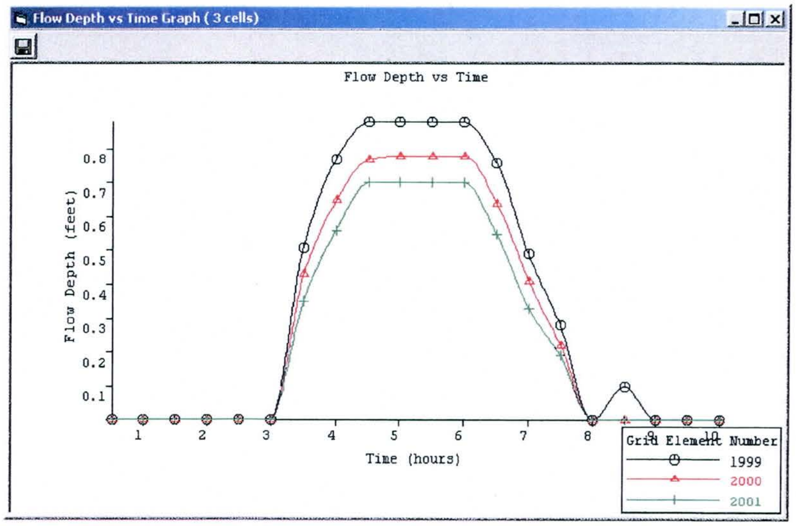


Click on any grid element to select it and then right click to show the following dialog box. Click either the *Flow Depth vs. Time* or *Velocity vs. Time* plot.

The resultant flow depth plot as a function of time (output interval) for grid element number 2107 is shown below.



Plotting several grid elements at once is possible. Click *Tools/Time Plots* and click several grid elements. Right click on one of the selected elements and click *Flow Depth vs. Time*.



Deselect the grid elements by clicking the selected cell and click *Delete Cell Group*.

2b. Flood Damage Assessment Tool

Overview

Total flood damages or individual structure flood damage costs can be automatically computed in the MAPPER program. Computing flood damage costs is a direct means of evaluating flood risk. The flood damage assessment method is similar in approach to that applied by the U.S. Army Corps of Engineers. It requires a table of individual structure damages as a function of flow depth. The structure data has to be collected from field inspection and the structure footprint has to be outline in GIS software. This lesson shows how to perform a flood cost assessment based on the FLO-2D predicted maximum flow depths and the structure damage field data.

Required Data

The FLO-2D data files *.DAT and output files *.OUT including the housing shape and damage code table found in the *Damage Assessment Example* subdirectory are used for this lesson.

File	Content	Location*
FPLAIN.DAT	Digital Terrain Model file	*\Example Projects\Damage Assessment Example*
DamageCode_08.tbl	Table file	*\Example Projects\Damage Assessment Example*
Houses.shp	Shape file	*\Example Projects\Damage Assessment Example*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

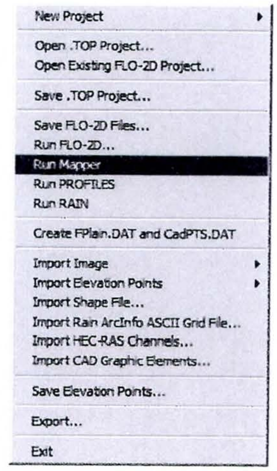
Step-by-Step Procedure

The following steps are necessary to conduct the flood damage assessment.

- Open MAPPER and load *Damage Assessment Example*;
- Damage assessment structure data discussion;
- Plot the FLO-2D predicted maximum flow depths;
- Import the building polygon shape file
- Compute the flood damage costs

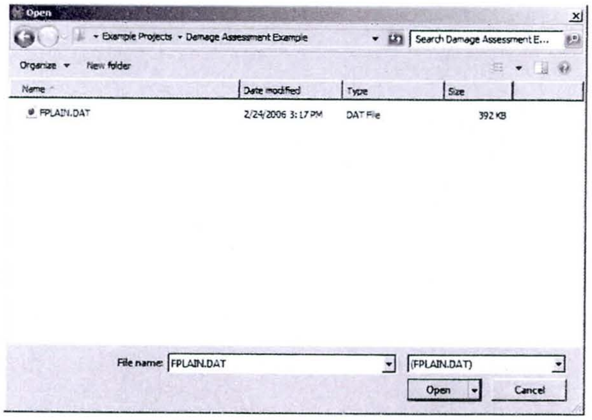
Step 1: Open MAPPER and load the Damage Assessment Example

In the GDS, select the *File* menu from the toolbar and click *Run MAPPER.*



Open the project output files by clicking on *File* and *Read FLO-2D Results...* as shown below.

Locate the project file folder and click on the *Open* button.



Step 2: Damage assessment structure data discussion

A table file (*.TBL) must be developed that contains the structure damage ($) as function of flood depth for each building type in the polygon shape file (DamageCode_08.tbl). This table is the result of intensive field inventory of the buildings in the potential area of inundation for the design flood event. The table file is organized as follows. This table is generated for a typical Corps of Engineers damage assessment.

Column 1: A damage code or number that corresponds to the polygon ID's in the shape file.

Columns 2 to 11: Cost associated with the maximum flow depth list in row 1 (depths in the following example range from 1 to 10 feet.

> NOTE: The damage table file (DamageCode_08.tbl) is hypothetical. The user should prepare an appropriate file adapted to each particular site.

```
DamageCode,1,2,3,4,5,6,7,8,9,10
1,5000,10000,15000,20000,25900,30500,35000,40000,45000,50000
2,3000,10000,15500,20500,25000,30300,35000,40000,45000,50000
3,4000,10500,15000,20300,25000,28000,35000,40000,45000,50000
4,5500,12000,15000,20000,25600,30000,35000,40000,45000,50000
5,5000,10000,15000,20000,25000,30000,35000,40000,45000,50000
```

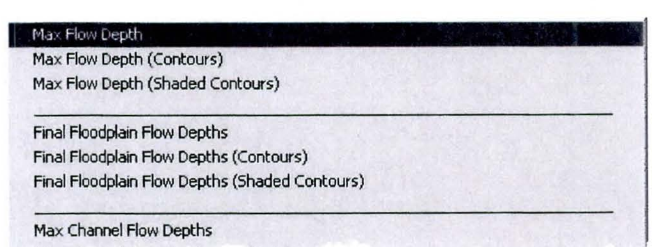
Line 1 indicates the name of each column. In this case it is depth in ft. Line 2 indicates that for polygon #1, 1 ft of flood depth will have a damage cost of $5000, 2 ft of flood depth will have a damage cost of $10,000, 3 ft depth will have a damage cost of $15000, and so on.

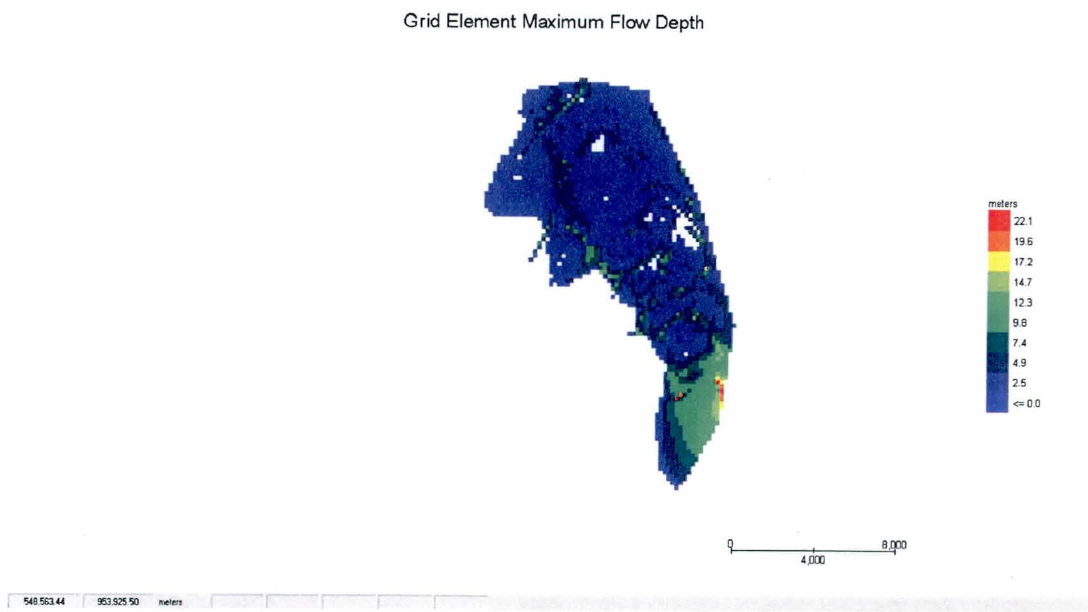
Another example damage table that used for agricultural field damages is show below. For this case, the damage flow depths are non-uniform.

```
DamageCode,0.2,0.3,0.4,0.5,0.6,1.0,1.1,1.5,1.6,10.
1,0,2908,5884,8838,14700,14700,16926,16926,17926,17926
2,0,755,1528,2295,3817,3817,4395,4395,4655,4655
3,0,1360,1360,1360,1360,1360,1360,1360,1360,1360
```

Step 3: Plot the FLO-2D predicted maximum flow depths

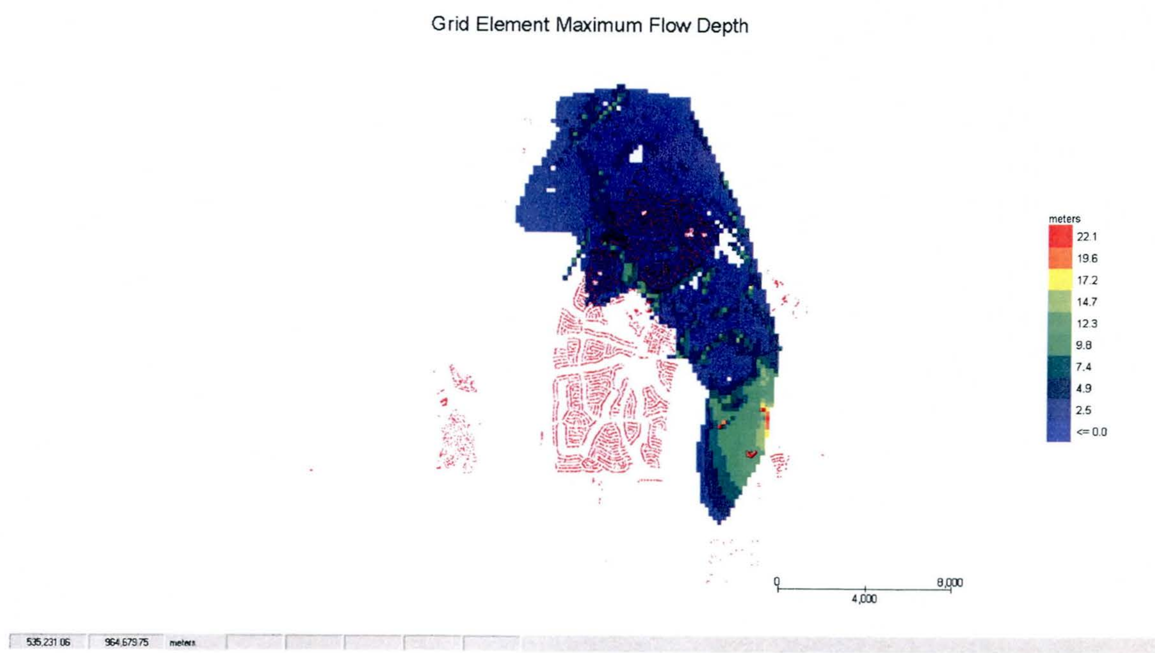
Click the *Max Flow Depth* command.



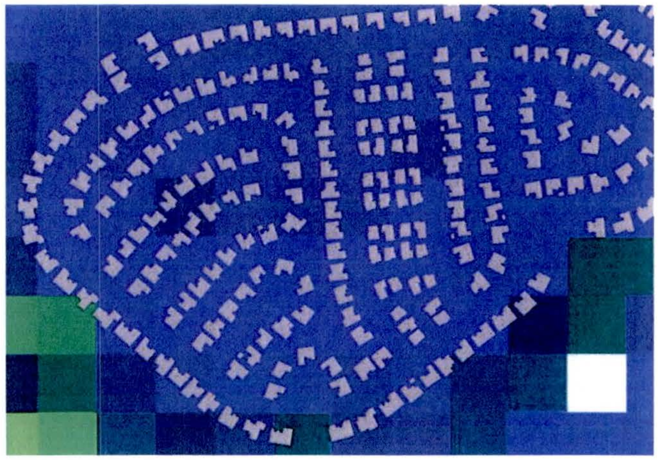


Step 4: Import the building polygon shape file

Import the building polygon shape file (*houses.shp*) using the *Import Shape file* command in the *File* menu. The housing shape file is simply a file of polygons of each house in a neighborhood. Each house polygon has ID, area, perimeter and damage code attributes.

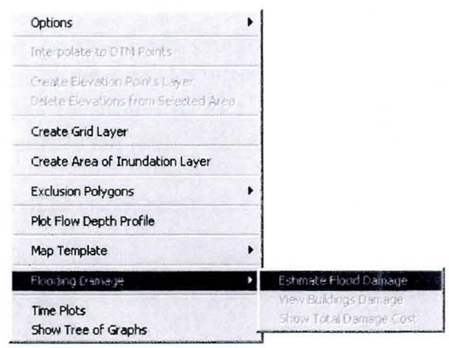


The following figure is a zoom view of the building shape file displayed over the grid element maximum flow depth plot in MAPPER.

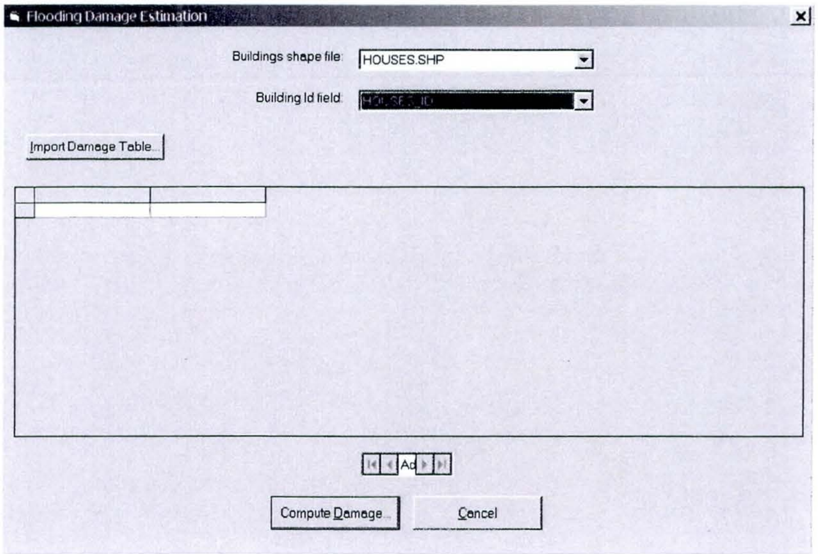


Step 5: Compute the flood damage costs

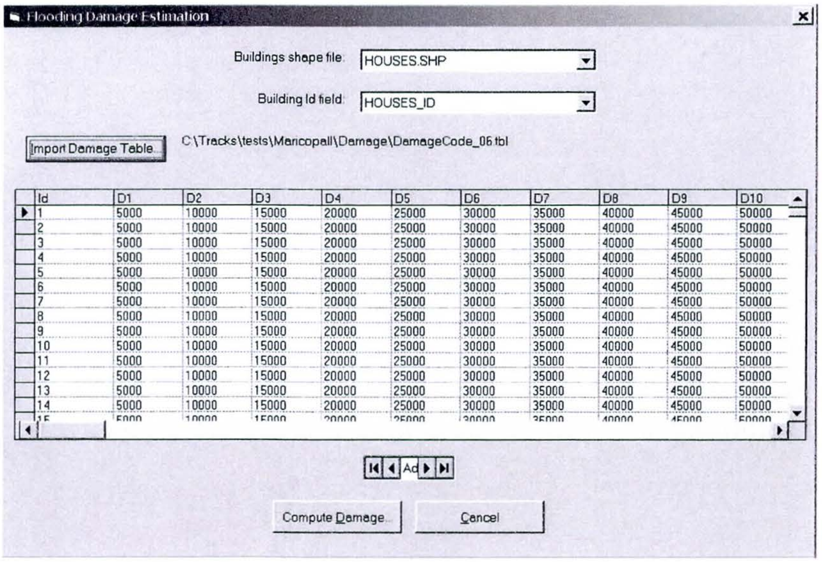
Select the *Estimate Flood Damage* command in the *Tools* menu.



The buildings shape file is named Houses.shp. The following dialog appears.



Select *HOUSES.SHP* from the *Buildings shape file* list and *HOUSES_ID* from the *Building Id field*. Import the damage table file *Damage-Code_08.tbl* using the button *Import Damage Table*. The table will be filled as shown bellow.

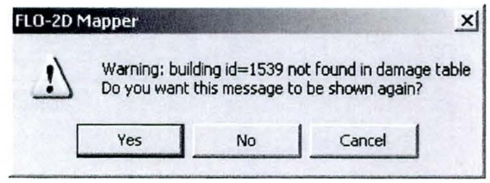


Id	D1	D2	D3	D4	D5	D6	D7	D8	D9	D10
1	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
2	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
3	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
4	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
5	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
6	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
7	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
8	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
9	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
10	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
11	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
12	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
13	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000
14	5000	10000	15000	20000	25000	30000	35000	40000	45000	50000

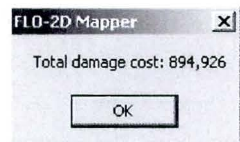
Click on *Compute Damage* button to calculate the total flood damage and to compute a damage cost estimate for each building. MAPPER will create a layer containing the building cost for each building intercepted by the flooding area.

NOTE: This computation make take several minutes depending on the number of buildings, the number of grid elements and computer speed.

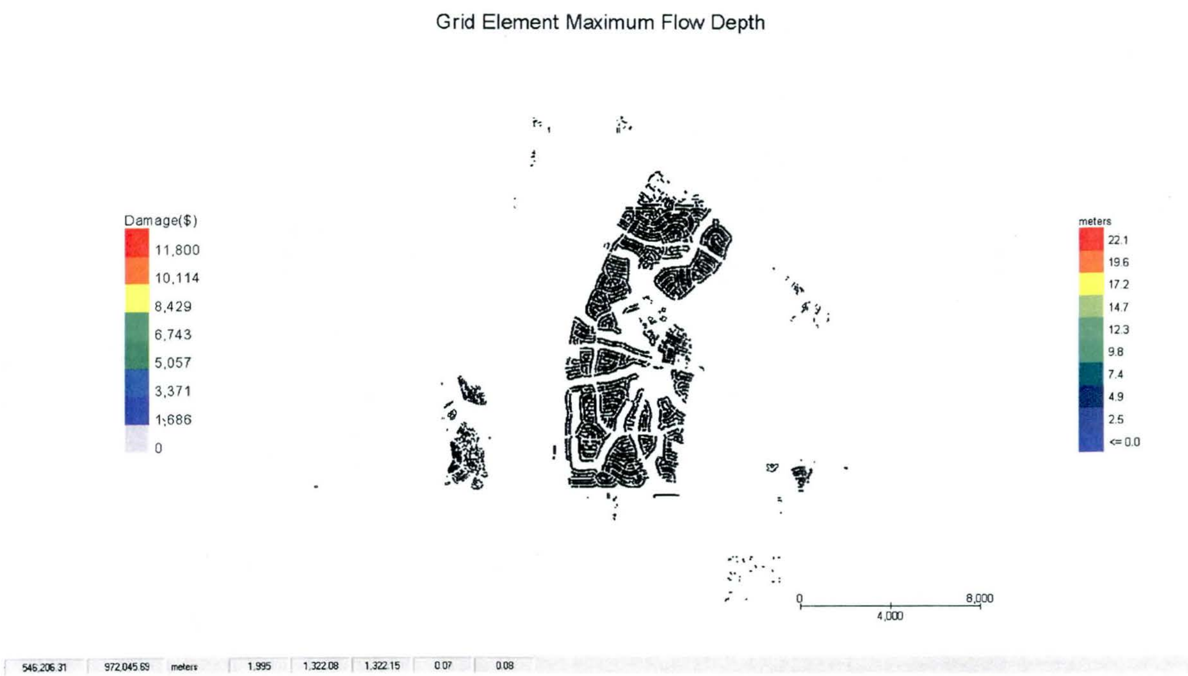
There may be buildings for which there is no damage value associated and the following message will appear.



Click *No* so the message will not appear for other buildings and continue the calculation. The total damage cost estimate for all the buildings will be displayed.

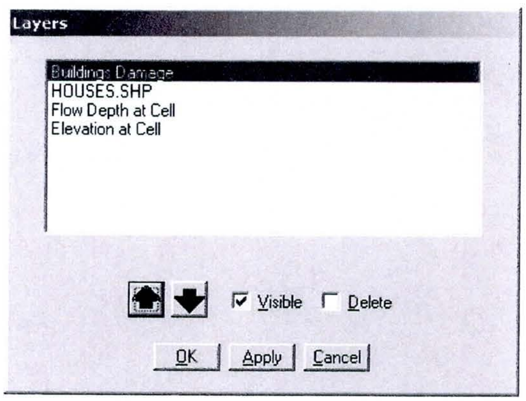


A color will be assigned for each building as function of the damage cost.

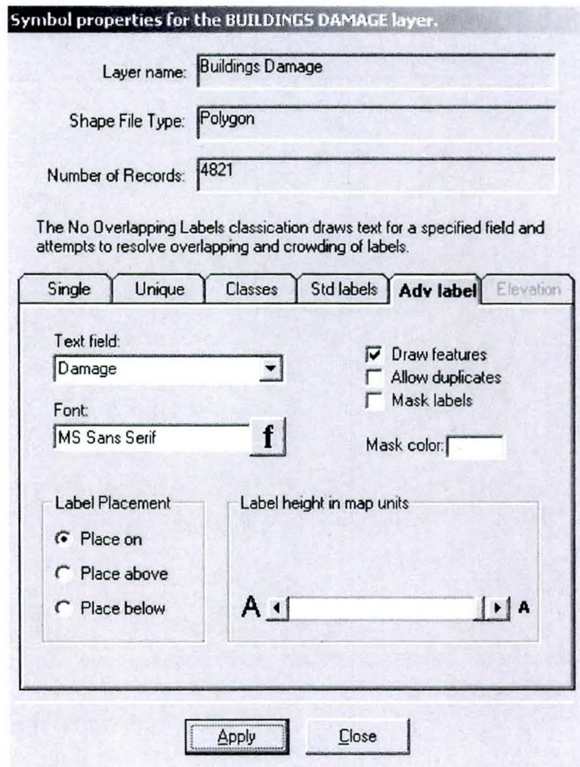


The building damage detail by color can be seen by 'zooming in' as shown.

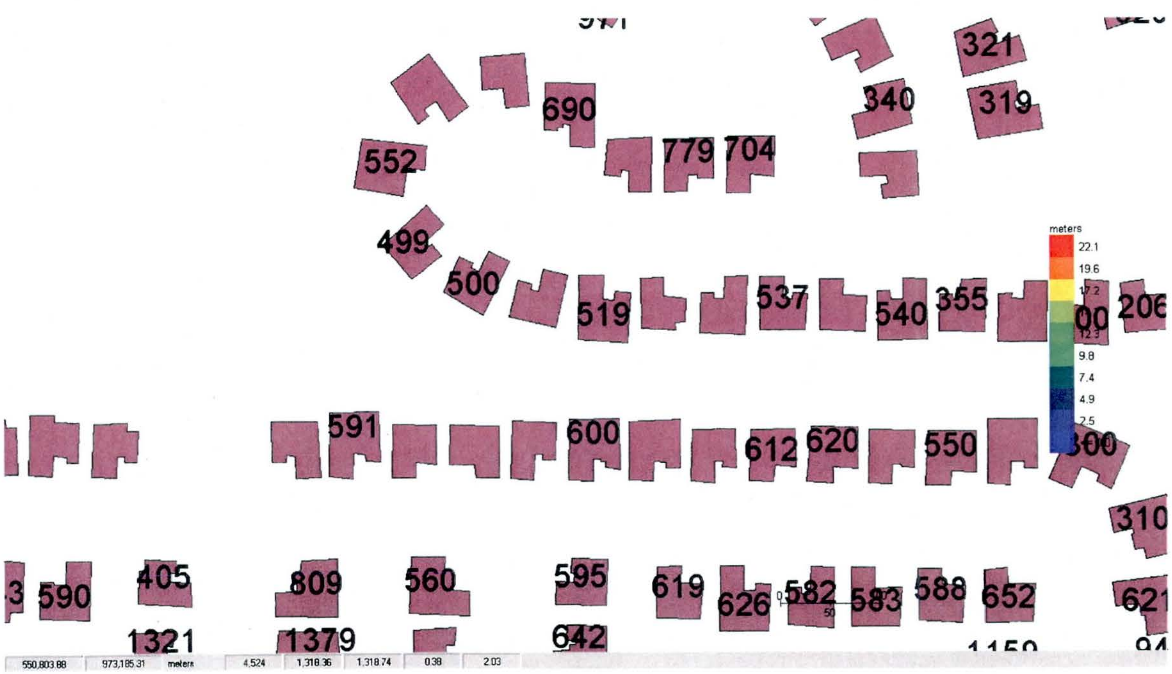
To view the individual building damage cost estimates click on the *View/Layers List* and double click on the *Building Damage* layer.



The *Symbol Properties for the BUILDINGS DAMAGE Layer* dialog box will appear. Select the *Adv label* option and adjust the font size by moving the *Label height* in map units control and clicking on the *Apply* button until the desired font size is achieved.



Click on the *Close* button when finished adjusting the building damage number text size. The following figure displays the damage cost layer indicating the damage estimate in $ (currency) for each building.



2c. Flood Hazard Mapping Tool

Overview

Hazard maps are a function of both flood intensity and probability. Flood intensity is defined by the flow depth and velocity. Flood probability is inversely related to flood magnitude. Flood hazard is then defined s a discrete combined function of the event intensity and return period. This lesson describes how to create a flood hazard map using the MAPPER program.

Required Data

This lesson requires the FLO-2D data files *.DAT and output files *.OUT found in the *Lesson 6* subdirectory

File	Content	Location*
FPLAIN.DAT	Digital Terrain Model file	*\Example Projects\Lesson 6*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

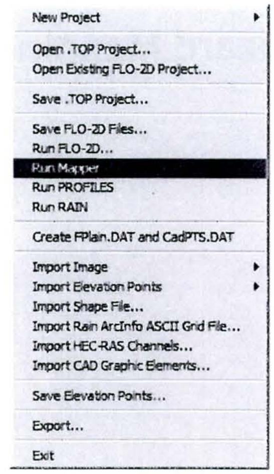
Step-by-Step Procedure

The following steps are necessary to conduct the flood damage assess-ment.

- Open MAPPER and load *Lesson 6*;
- Import aerial image;
- Plot hazard map;
- Plot hazard map (shaded contour).

Step 1: Open MAPPER and load the Lesson 6

In the GDS, select the *File* menu from the toolbar and click *Run MAPPER.*

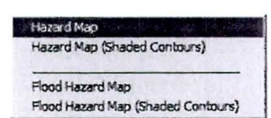


Open the project output files by clicking on *File* and *Read FLO-2D Results…* as shown below.

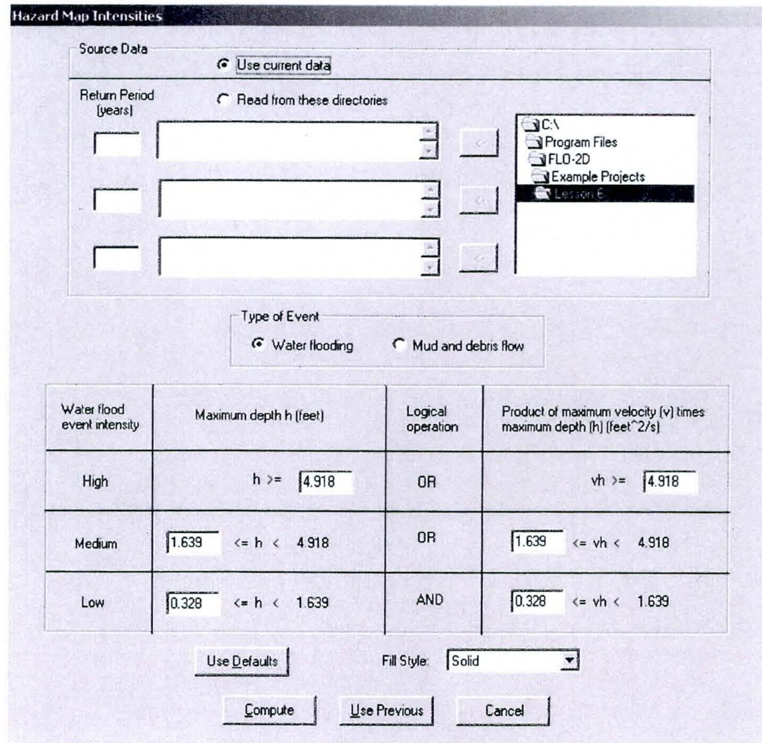
Step 2: Load aerial image (See Lesson 1, Step 4)

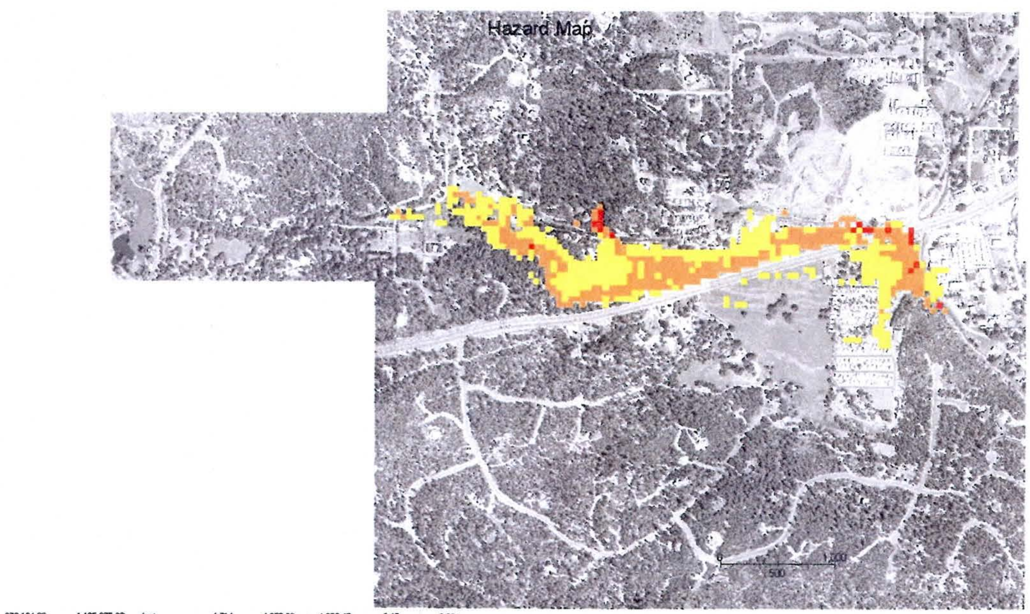
Step 3: Plot hazard map

Click the *Hazard Map* button and click *Hazard Map*.



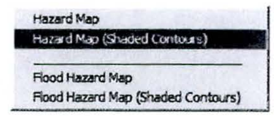
The *Hazard Map Intensities* dialog box appears. This box has two mapping options. Use the default intensity, depth and velocity parameters for water flooding on the current data. Click *Compute* to generate the hazard map.

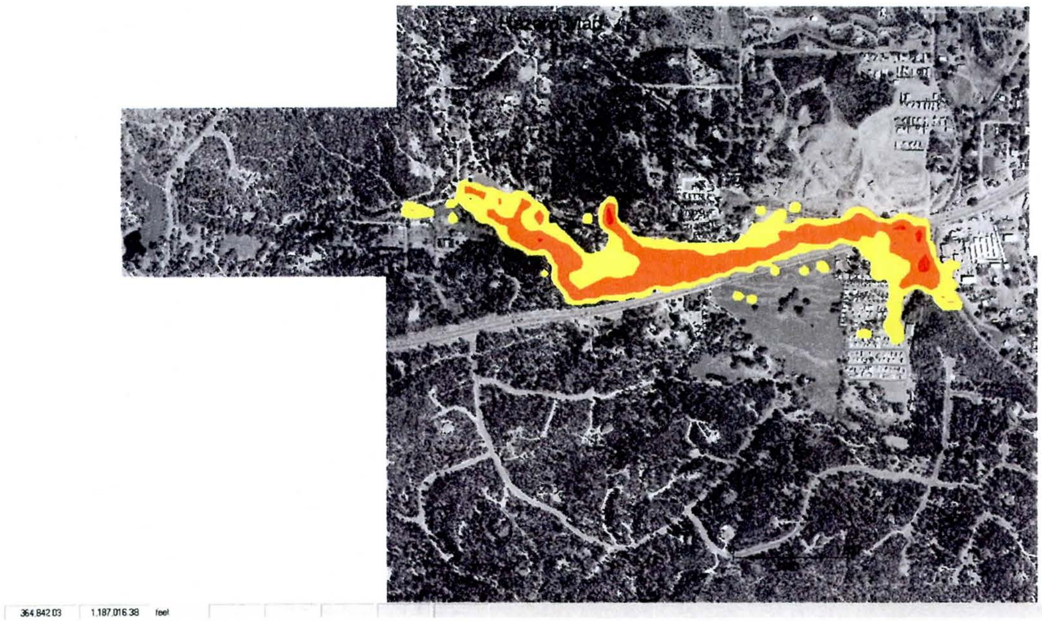




Step 4: Plot hazard map (Shaded Contour)

Click the *Hazard Map* button and click *Hazard Map*.





Summary

In this Lesson, the MAPPER program has been used to graphically display and analyze the FLO-2D output data. MAPPER is a versatile mapping tool that can be used to create high resolution flood hazard maps. MAPPER automatically generates shape files of each flood plot that can imported into another GIS or CADD mapping program. Also use MAPPER for more advanced applications such as making movies of a flood simulation and conducting flood damage assessments. The movies can be helpful tools for project flood presentations to view predicted floodwave progression. The flood damage assessments provide a valid estimate of flood risk.

Review Questions

1. When are the MAPPER shape files created?

   ____

   ____

2. How are aerial images geo-referenced with the grid system coordinates? ____________

   ____

3. To work with the DTM points in MAPPER, what must be created first?

   ____

4. Contour plots can be created for these six flow depth results.
   1. ____
   2. ____
   3. ____
   4. ____
   5. ____
   6. ____

5. What is the difference between maximum flow depth and final flow depth results? _____

   ____

6. Outflow grid elements are included in the flow depth results? (True or False) _________

7. DTM ground surface points are subtracted from the predicted maximum water surface elevation for what purpose? ____

8. Why are grid element elements used instead of DTM points to compute flood hydraulics?

   ____

   ____

9. Exclusion polygons are used for what purpose? ______________________________

   ____

10. What tools are used to improve shaded contour plots?
    1. ____
    2. ____
    3. ____
    4. ____

5. ____

11. MAPPER flood animation requires what controls to be set prior to the flood simulation?

____

12. For the flood damage assessment, how are the building damage tables and polygon shape files developed? ____

____

13. The total damage cost for a design flood event is a measure of flood risk? (True or False)? ____________

14. Flood hazard maps represent an assessment of what two flood factors? ____________

____

15. In MAPPER, flood hazard maps are based on maximum depth and maximum product of depth times velocity squared? (True or False) ____________

LESSON 7 – RAINFALL AND INFILTRATION

Overview

Rainfall and infiltration parameters can be entered and edited using the GDS. The object of this tutorial is to use the GDS to create spatially variable rainfall parameters for the RAIN.DAT file and to work through the various methods to assign infiltration data. The project data presented is all that is necessary is to build the two data files. A rainfall distribution will be selected for rainfall model and the infiltration data will be entered both as global and spatially variable parameters.

Required Data

The profiles program requires five basic data files plus the infiltration file and the rainfall file. The five basic files are provided. These files are found in the folder *Lesson 7*.

- CONT.DAT
- CADPTS.DAT
- FPLAIN.DAT
- TOLER.DAT
- OUTFLOW.DAT
- RAIN.DAT (user made)
- INFIL.DAT (user made)

File	Content	Location*
**.dat*	Grid system data files	*\Example Projects\Lesson 7/GreenAmpt*
	Rainfall distribution data	*Built In*
LandUse.shp	Green-Ampt shape file	*\Example Projects\Lesson 7/GreenAmpt*
Soil.shp	Green-Ampt shape file	*\Example Projects\Lesson 7/GreenAmpt*
LandUse.tlb	Green-Ampt table file	*\Example Projects\Lesson 7/GreenAmpt*
Soil.tlb	Green-Ampt table file	*\Example Projects\Lesson 7/GreenAmpt*
**.dat*	Grid system data files	*\Example Projects\Lesson 7/SCS*
Land.shp	SCS shape file	*\Example Projects\Lesson 7/SCS*
Soil.shp	SCS shape file	*\Example Projects\Lesson 7/SCS*
Imperv.shp	SCS shape file	*\Example Projects\Lesson 7/SCS*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To assign rainfall infiltration data with the GDS program follow these steps.

Input Rain

- Create a new project folder;
- Open the GDS and load the *Lesson 7* project;
- Create a rainfall distribution and data file;
- Save and run the model;

Set-up Default Green-Ampt Global Infiltration

- Set-up global Green-Ampt infiltration parameters;
- Save and run the model;

Green-Ampt Spatially Variable Infiltration

- Load infiltrations shape files;
- Load landuse and soil tables;
- Compute infiltration data file;
- Manually edit the INFIL.DAT file;
- Save and run the model;

SCS Curve Number Spatially Variable Infiltration

- Open project;
- Load infiltration shape files;
- compute infiltration;
- Save and run the model;

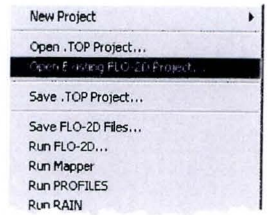
INPUT RAIN

Step 1: Create a new project folder

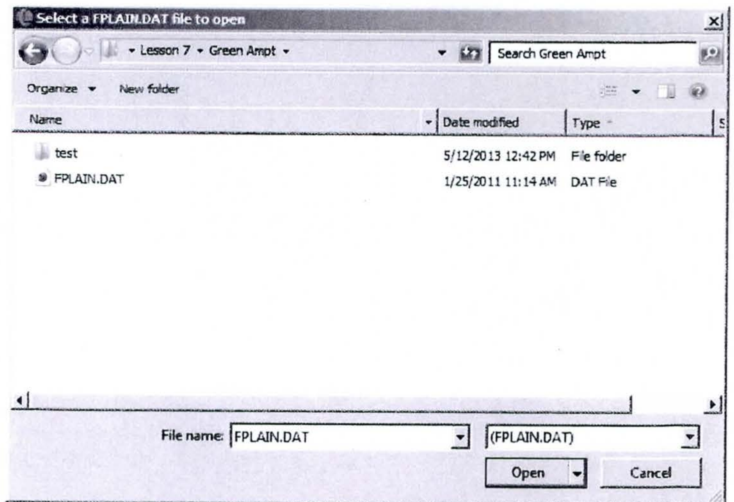
Several different project data sets are created in this lesson. It is advisable to setup a project folder when saving new project data so that a comparison with previous infiltration model is available. In an explorer window, copy the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation \Example Projects\Lesson \Green Ampt* and rename it to *Test*.

Step 2: Open the Grid Developer System program

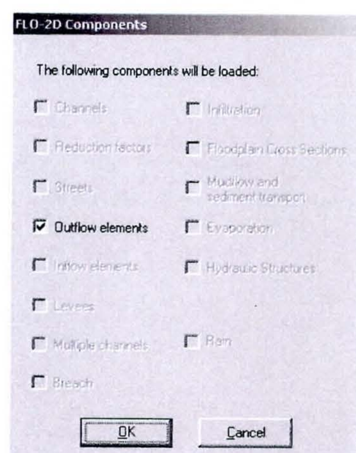
Double click the GDS Basic icon from the desktop. 



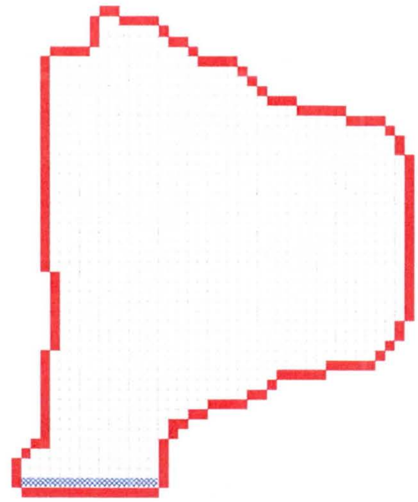
Browse the folder *C:\Users\Public\Documents\FLO-2D Basic Documentation \Example Projects\Lesson 7\Green Ampt* and double click the file *FPLAIN.DAT*.



The following dialog box will be displayed. Click *OK* to load the model and display the grid system.

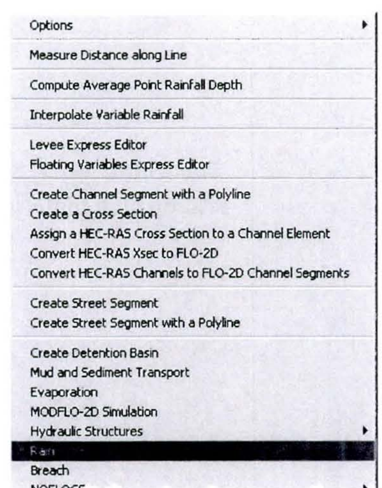


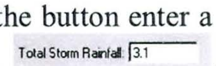
The display of the grid system should look like the following image.



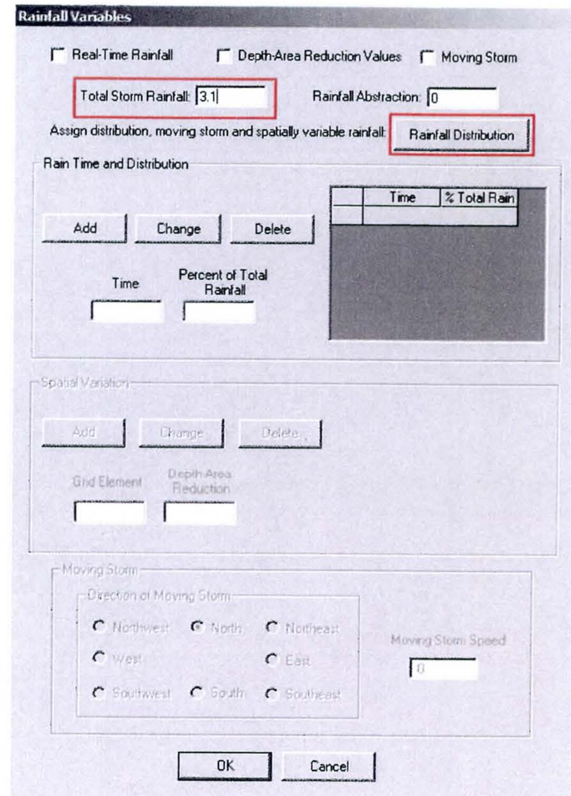
Step 3: Set-up total rainfall and a rainfall distribution

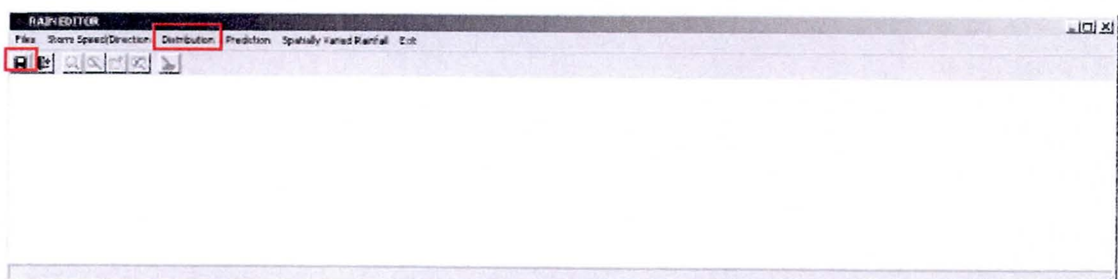
To create a rainfall distribution, click the *Rain* command on the *Tools* Menu.



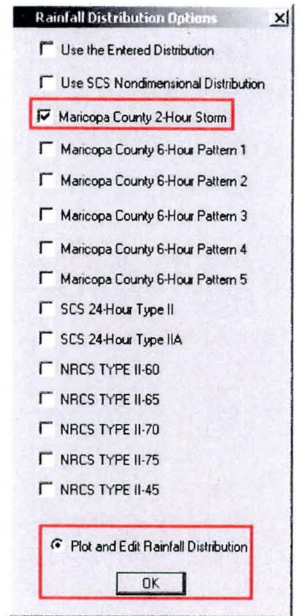
Initially the *Rainfall Distribution* button will be grayed . To activate the button enter a value of *3.1 inches* in the *Total Storm Rainfall* box 

. The *Rainfall Distribution* button is now active. Click it to open the *Rain Editor* dialog box.

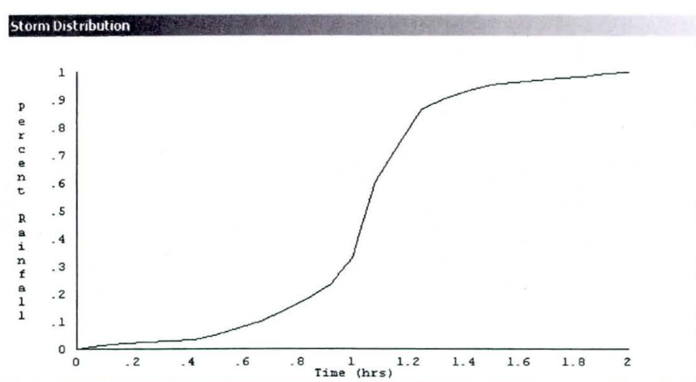


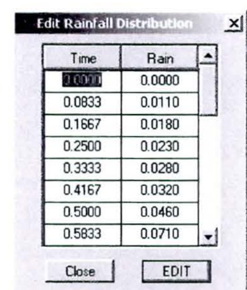


Click the *Distribution* button and select the *Maricopa County 2-Hour Storm* option and click the *Plot and Edit Rainfall Distribution* radio button and click *OK*.



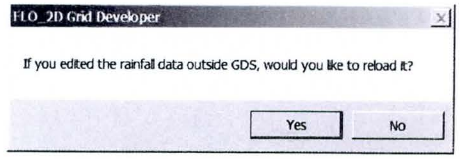
The *Storm Distribution* plot and *Edit Rainfall Distribution* dialog box appear. Click *Close* and then click the *Save* button on the *Rain Editor* menu bar.

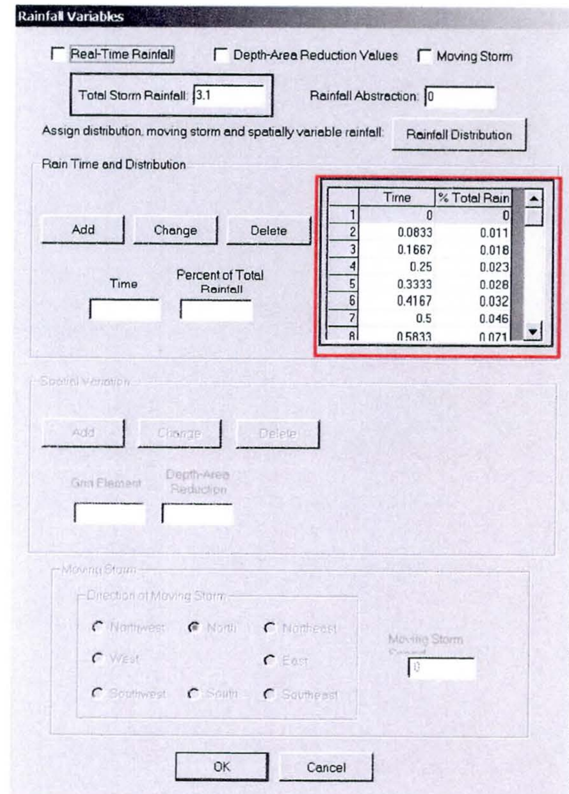




The *RAIN.DAT* file has just been created. Click *OK* and *exit* the *Rain Editor* dialog box.

Return to the GDS and click *Yes* to load the rain data table.



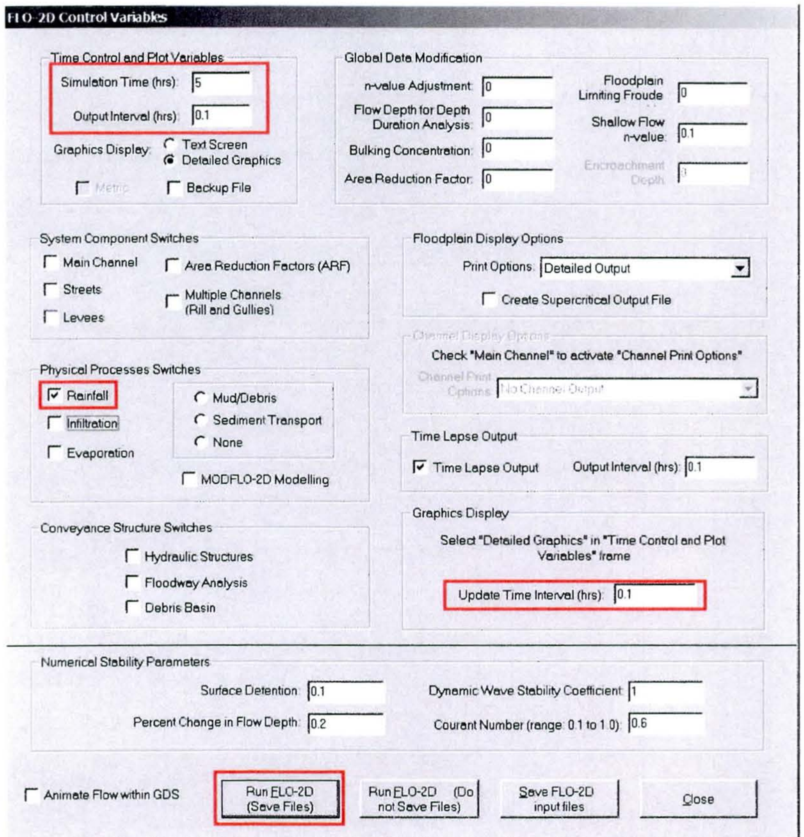


The data is loaded into the table click *OK*.

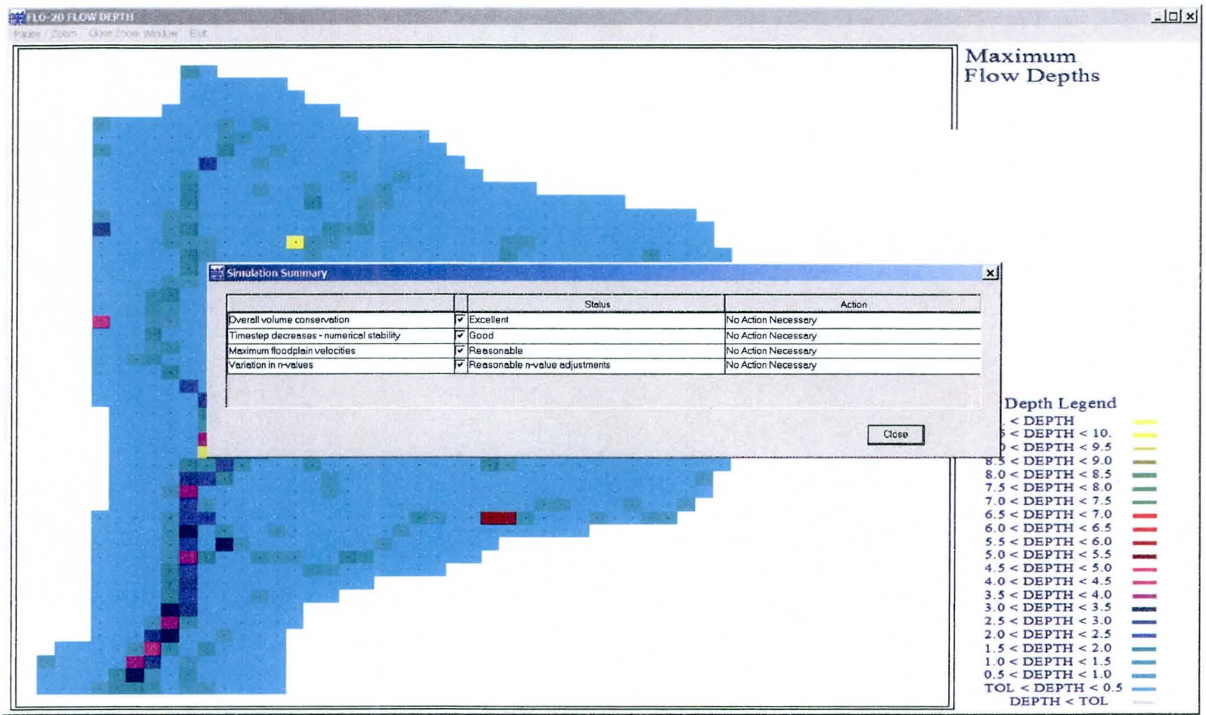
Step 4: Save the data files and run the model

Save the FLO-2D data files. To run the FLO-2D flood simulation, use the *Run FLO-2D…* command in the *File* menu.

Make sure all the correct component switches are turned on (Rain). Input the *Simulation Time* = 5 hr, *Output Interval* = 0.1 hr for more output intervals, select the *Detailed Graphics* option and enter an *Update Time Interval* of 0.1 hours as shown.



Click *Run FLO-2D* and observe the floodwave progression over the project area. The rainfall hydrograph will be plotted and volume conservation is reported during the simulation. Close the GDS.



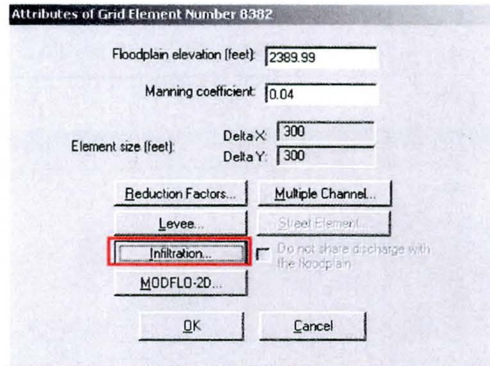
GREEN-AMPT GLOBAL INFILTRATION

Step 5: Create a new project folder

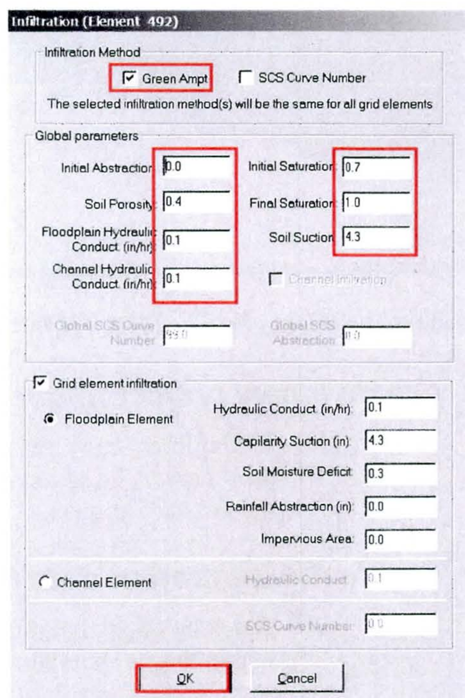
In Explorer, copy the folder *C:\Program Files\FLO-2D\Example Projects\Lesson 7* and rename it to *Lesson 7 Rain*. Open the GDS and load the *Lesson 7* project.

Step 6: Create the infiltration data file

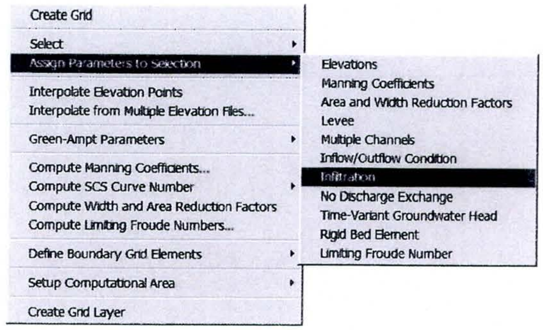
To set up the global infiltration values *double-click* any grid element within the computational boundary to bring up the *grid element attributes editor*. Click the *Infiltration* button.



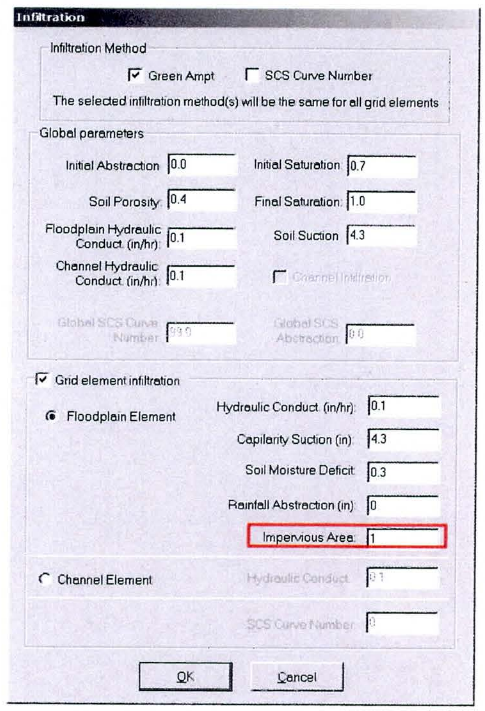
Fill out the Infiltration editor box as shown in the following image. Click *OK* twice to exit the editor boxes. Check the *Grid element infiltration* box. This action will cause the global infiltration values to be written to the INFIL.DAT file along with the values for *grid element 492*.



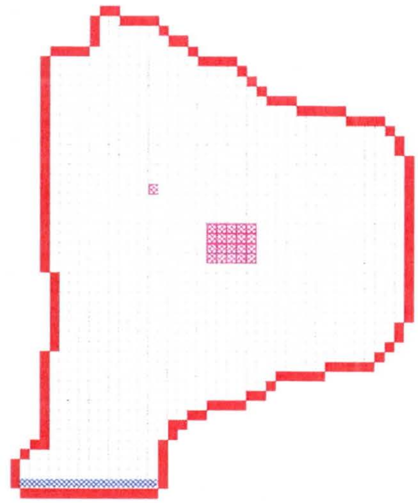
Turn off the infiltration for specific nodes by creating impervious elements. Click the *Select element by element* # button and select a group of neighboring grid elements. Click *Grid/Assign Parameters to Selection/Infiltration*.



Turn on the impervious area switch by entering the number *1* in the *Impervious Area* box and click *OK*.

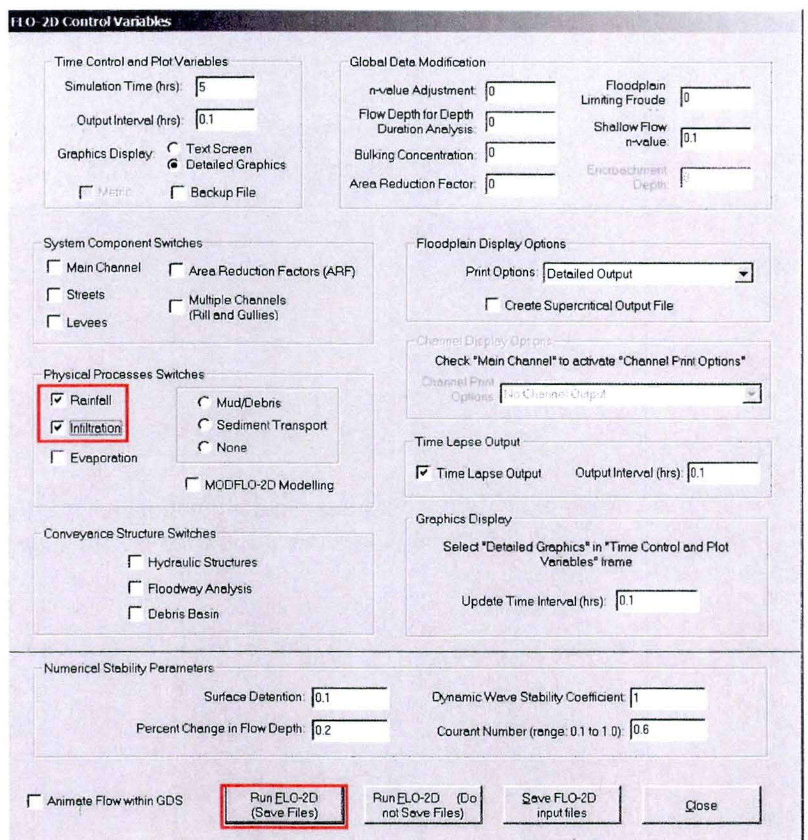


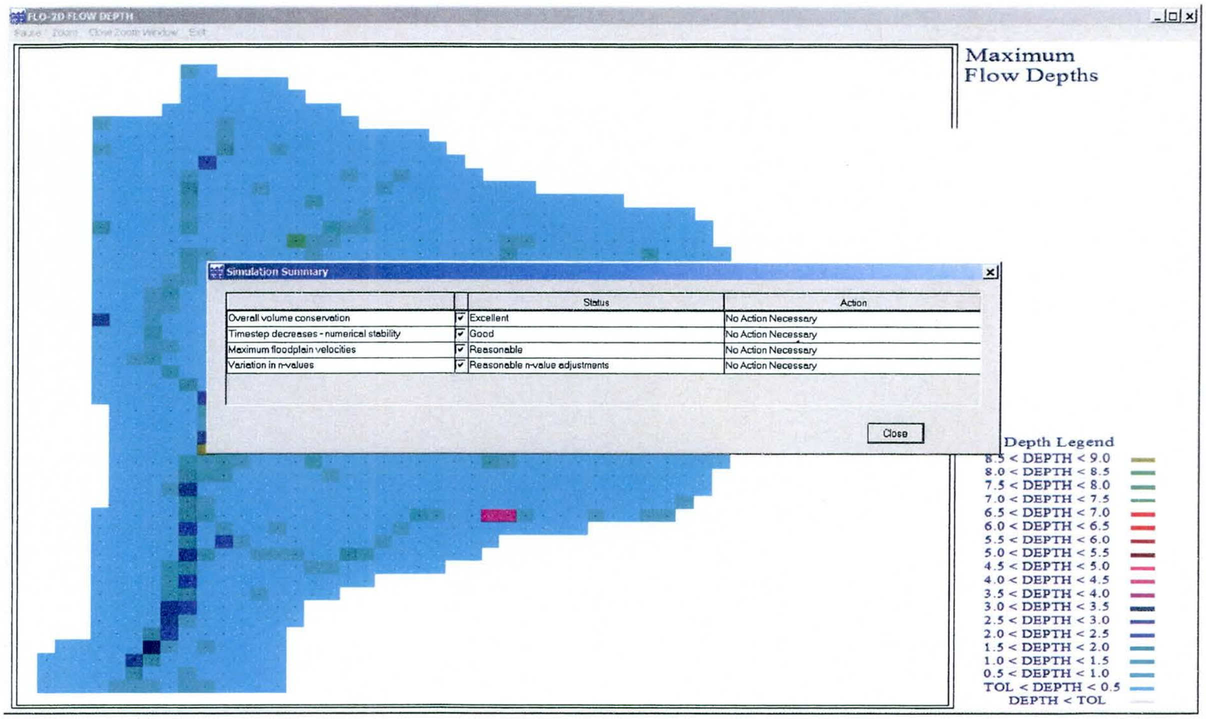
Note in the following image that while the global infiltration is turned on, only the spatially variable infiltration is shown on the grid system.



Step 7: Save the FLO-2D Files and run the model

Save the FLO-2D data files and run the Model. Click *File/Run FLO-2D...* and note the *Infiltration* switch is checked. Click *Run FLO-2D (Save Files)*.





A review of the SUMMARY.OUT file shows that the total infiltration loss is 1446.36 Acre-ft. Close the GDS.

```
=====================================================================
                          *** OUTFLOW (ACRE-FT) ***

OVERLAND INFILTRATED AND INTERCEPTED WATER      0.62 INCHES


          OVERLAND FLOW                            WATER

WATER LOST TO INFILTRATION & INTERCEPTION        1446.36

FLOODPLAIN STORAGE                               5134.93

FLOODPLAIN OUTFLOW HYDROGRAPH                     423.82
                                               ---------

FLOODPLAIN OUTFLOW, INFILTRATION & STORAGE       7005.11

=====================================================================
```

GREEN-AMPT INFILTRATION FROM SHAPE FILE

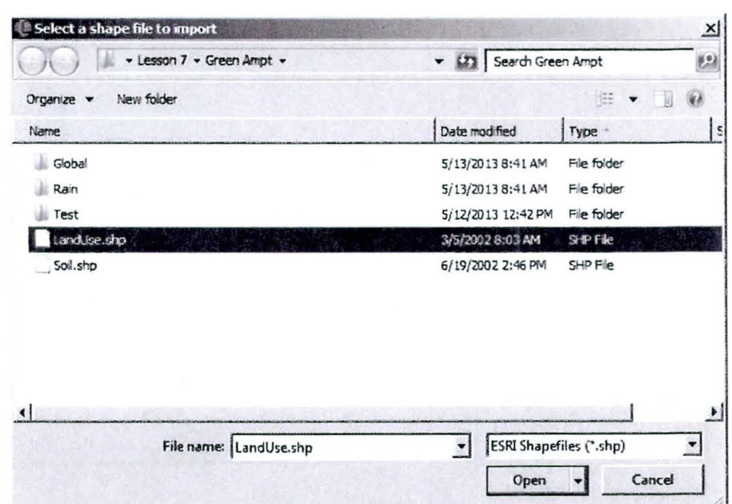
Step 8: Create a new project folder

In Explorer, copy the folder *C:\Program Files\FLO-2D\Example Projects\Lesson 7* and rename it to *Lesson 7 Global Green-Ampt*. Open the GDS and load the *Lesson 7* project.

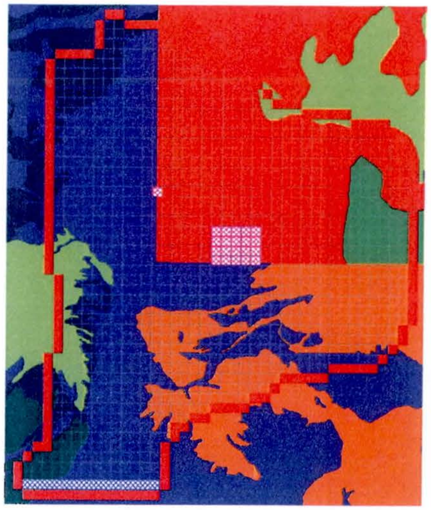
Step 9: Load Landuse and Soil shape files

Click *File/Import Shape File...*

Load the *Landuse* shape file found in *Lesson 7* and repeat the process for *Soil* shape file.

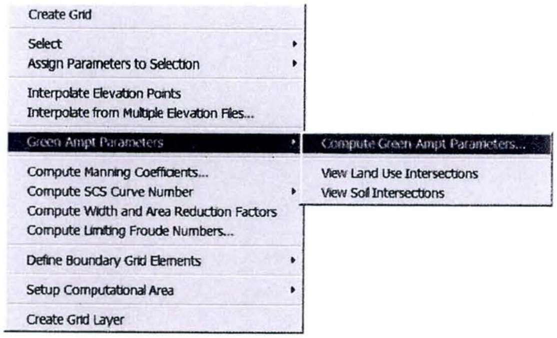


The shape files are shown in the following image.

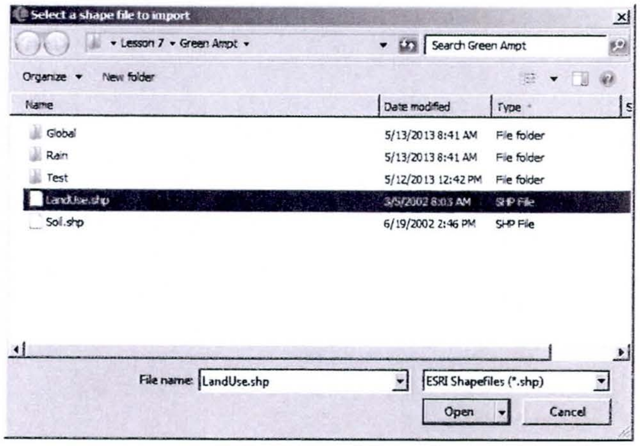


Step 10: Load Landuse and Soil tables;

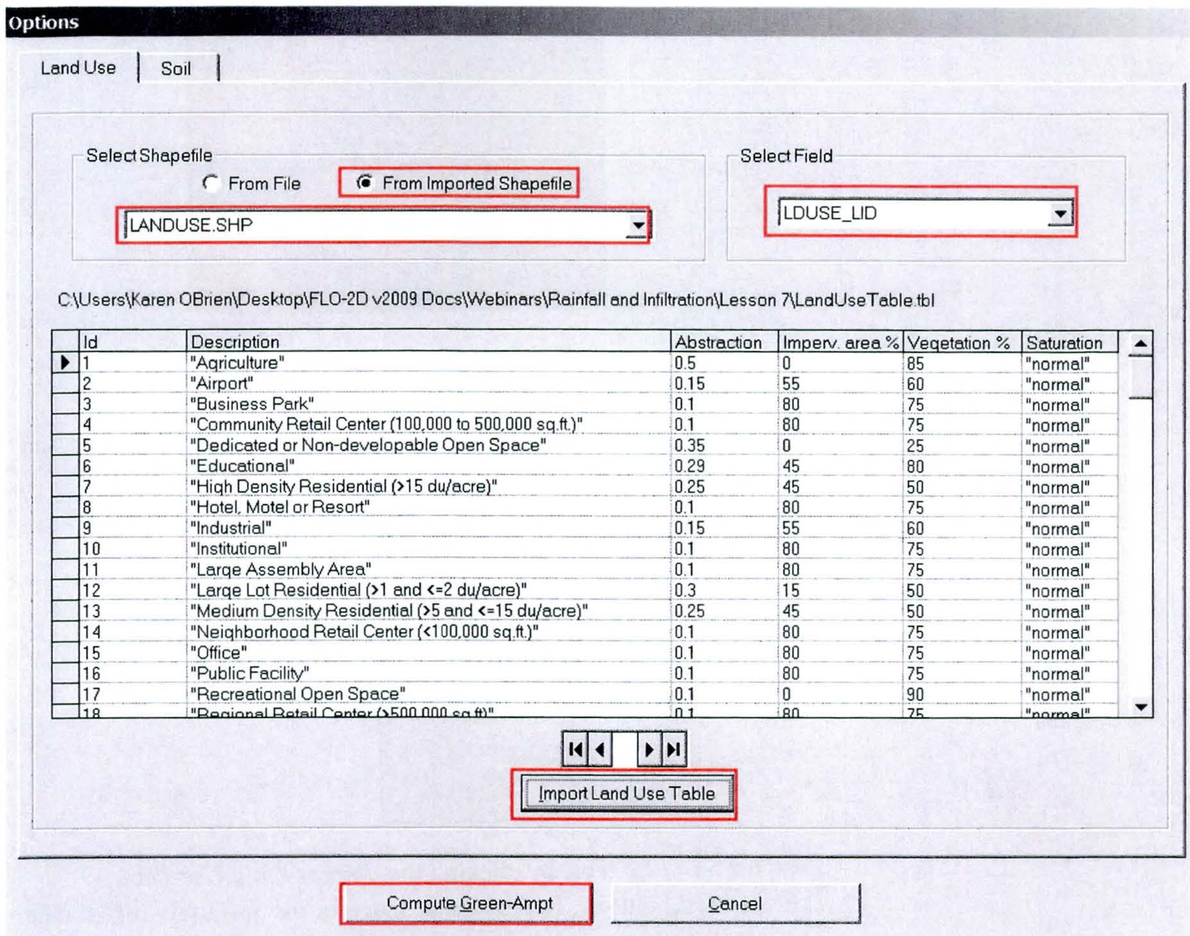
Click *Grid/Green-Ampt Parameters/Compute Green-Ampt Parameters...*



Load the *Landuse* table by clicking the *Import Land Use Tables* [Import Land Use Table] button. Repeat the process for the *Soil* table [Import Soil Table]. The landuse and soil tables are found in the *Lesson 7* folder.

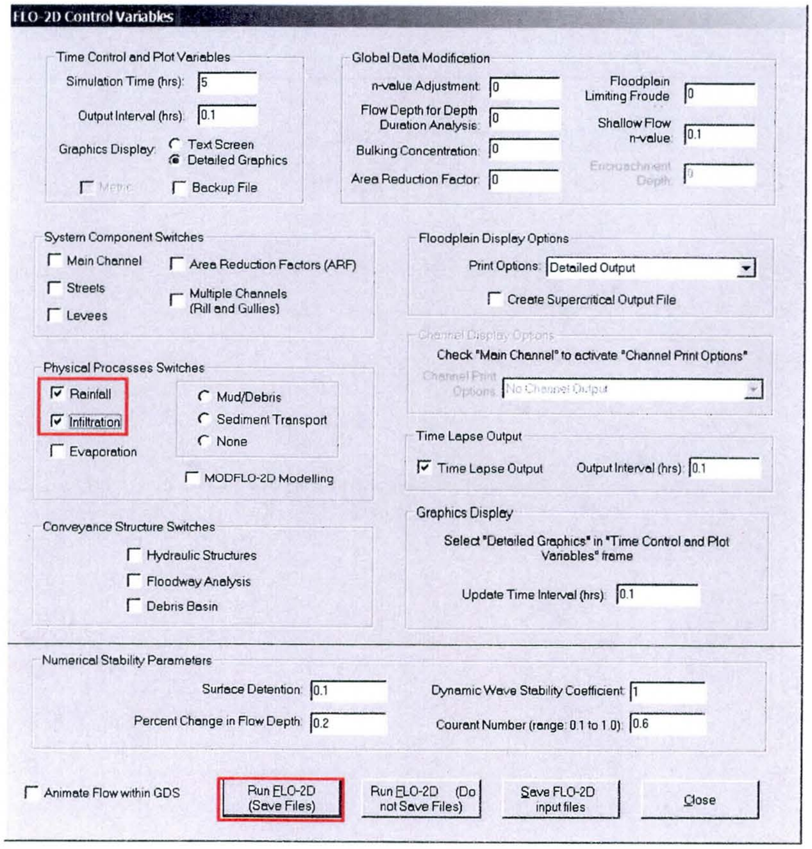


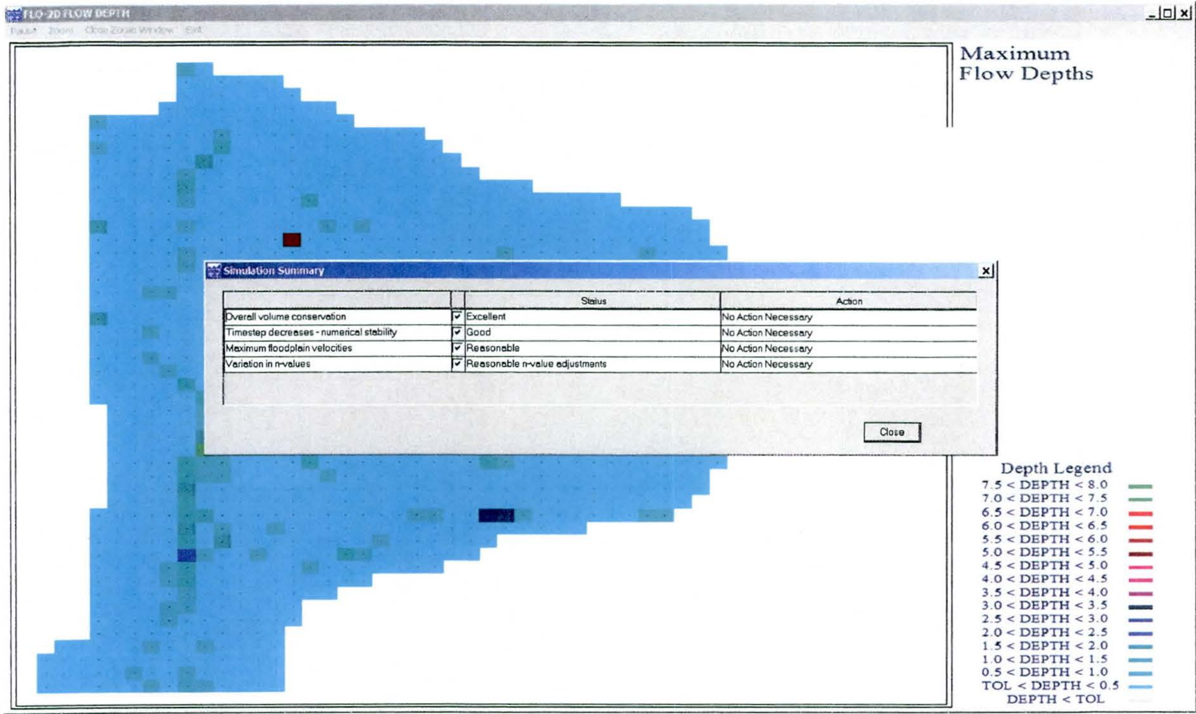
Click the *From Imported Shape file* radio button and select the shape file *LANDUSE.SHP*. Select *LDUSE_ID* as the field. Repeat the procedure for the *Soil* tab. Select *SOIL.SHP* as the shape file and *SOIL_LID* as the field. Click the *Compute Green-Ampt* button.



Step 11: Save the FLO-2D Files and run the model

Save the FLO-2D data files and run the Model. Click *File/Run FLO-2D...* and note the *Infiltration* switch is checked. Click *Run FLO-2D (Save Files)*.


FLO-2D Control Variables
Time Control and Plot Variables
Simulation Time (hrs): 5
Output Interval (hrs): 0.1
Graphics Display: Text Screen
Detailed Graphics
Metric
Backup File
Global Data Modification
n-value Adjustment: 0
Flow Depth for Depth Duration Analysis: 0
Bulking Concentration: 0
Area Reduction Factor: 0
Floodplain Limiting Froude: 0
Shallow Flow n-value: 0.1
Encroachment Depth: 0
System Component Switches
Main Channel
Streets
Levees
Area Reduction Factors (ARF)
Multiple Channels (Rill and Gullies)
Floodplain Display Options
Print Options: Detailed Output
Create Supercritical Output File
Channel Display Options
Check "Main Channel" to activate "Channel Print Options"
Channel Print Options: No Channel Output
Physical Processes Switches
Rainfall
Infiltration
Evaporation
Mud/Debris
Sediment Transport
None
MODFLO-2D Modelling
Time Lapse Output
Time Lapse Output
Output Interval (hrs): 0.1
Conveyance Structure Switches
Hydraulic Structures
Floodway Analysis
Debris Basin
Graphics Display
Select "Detailed Graphics" in "Time Control and Plot Variables" frame
Update Time Interval (hrs): 0.1
Numerical Stability Parameters
Surface Detention: 0.1
Percent Change in Flow Depth: 0.2
Dynamic Wave Stability Coefficient: 1
Courant Number (range: 0.1 to 1.0): 0.6
Animate Flow within GDS
Run FLO-2D (Save Files)
Run FLO-2D (Do not Save Files)
Save FLO-2D input files
Close



A review of the SUMMARY.OUT file shows that the total infiltration loss is 2298.57 Acre-ft. Close the GDS.

```
================================================================
                          *** OUTFLOW (ACRE-FT) ***

OVERLAND INFILTRATED AND INTERCEPTED WATER        1.10 INCHES


          OVERLAND FLOW                             WATER

WATER LOST TO INFILTRATION & INTERCEPTION          2298.57

FLOODPLAIN STORAGE                                 4435.56

FLOODPLAIN OUTFLOW HYDROGRAPH                       264.68
                                                 ---------

FLOODPLAIN OUTFLOW, INFILTRATION & STORAGE         6998.81

================================================================
```

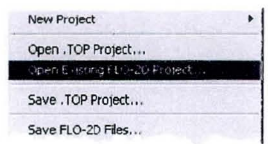
SCS Curve Number Spatially Variable Infiltration

Step 12: Open the GDS and load project

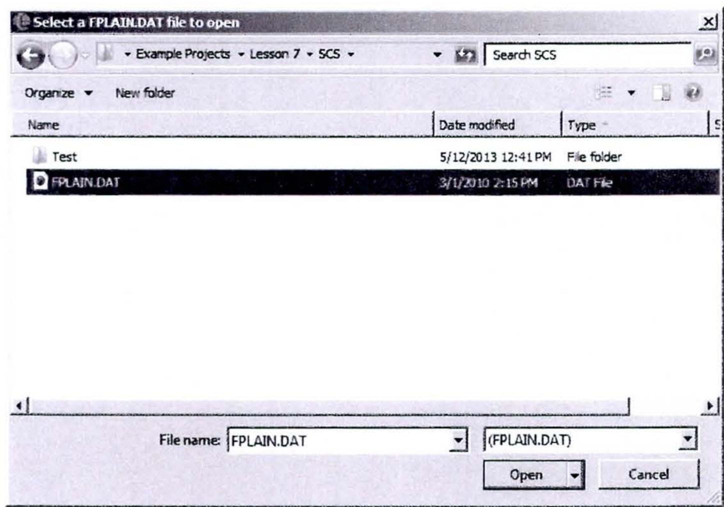
NOTE Using an Explorer window, make a new folder in the Lesson 7/SCS folder named "TEST". Copy all the Lesson 7/SCS files to this folder and perform the lesson in the TEST folder. ***NOTE***

Double click the GDS Basic icon from the desktop. 

From the GDS *File* menu click *Open Existing FLO-2D Project...*

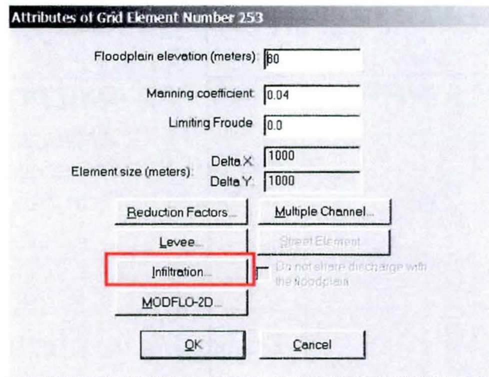


Locate the folder *Lesson 7/SCS* and open the *FPLAIN.DAT* file. Click *Open* and *OK* to load the project.

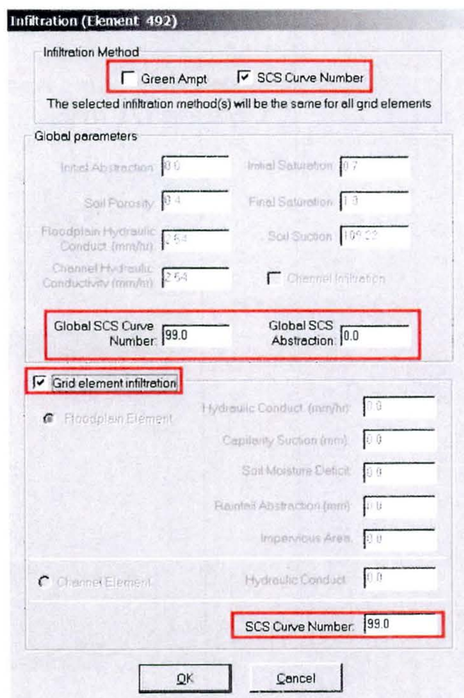


Step 13: Assign a global SCS curve number

Double click any grid element to load the *Attributes Editor* window and click *Infiltration*.



Turn off the *Green-Ampt* global infiltration by clicking the check box and turn on the *SCS Curve Number* global values by clicking the *SCS* check box. Click the *Grid Element Infiltration* for element *492*. Fill in the *Global SCS Curve Number* and *Global SCS Abstraction* if necessary.

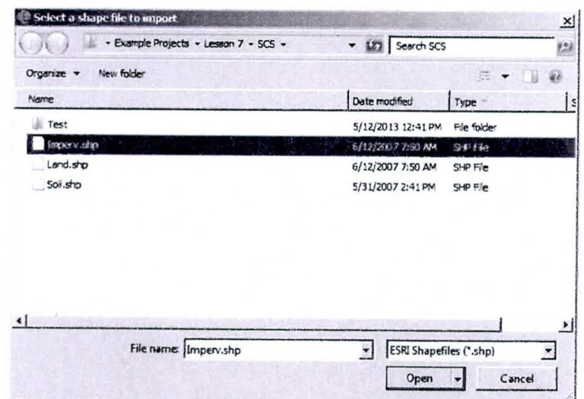


Click *OK* and click *File/Save FLO-2D Files...* The INFIL.DAT file is now located in the project folder.

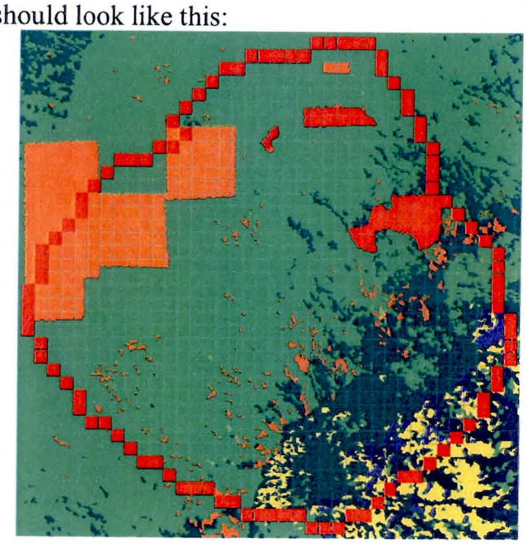
Step 14: Load infiltration shape files

Click the *File* menu and select *Import Shape File...*

Select the *Imperv.shp* file and click *Open*. Repeat the process for the *Land.shp* and *Soil.shp*.

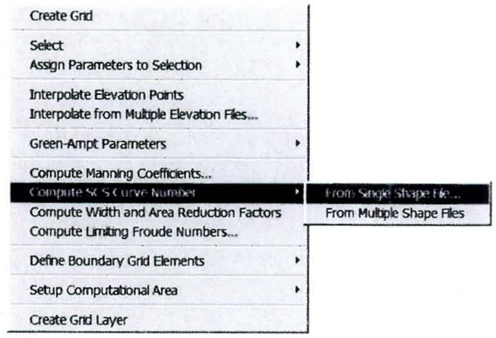


The project should look like this:

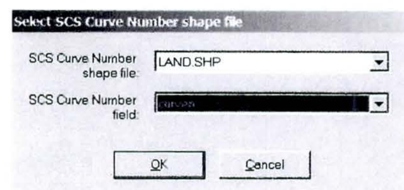


Step 15: Assign a SCS curve number from a single shape file

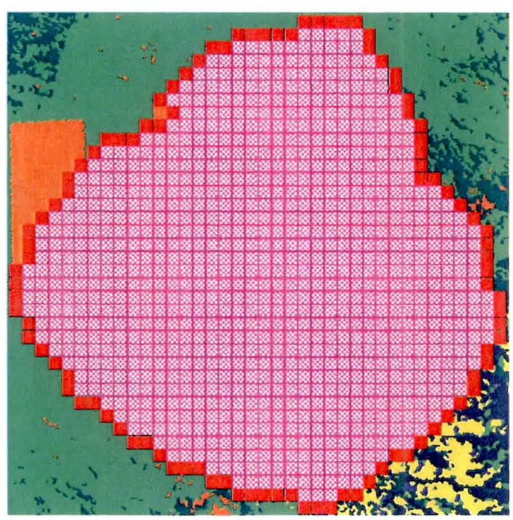
Click *Grid/Compute SCS Curve Number/From Single Shape File...*



Set the *SCS Curve Number shape file* to *LAND.SHP* and the field to *curven* and click *OK*.

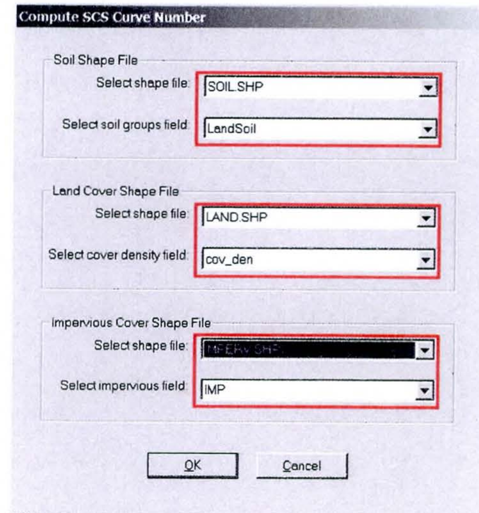


Click *File/Save FLO-2D Files...* The *INFIL.DAT* file now has spatially variable curve number data. The project should look like this:

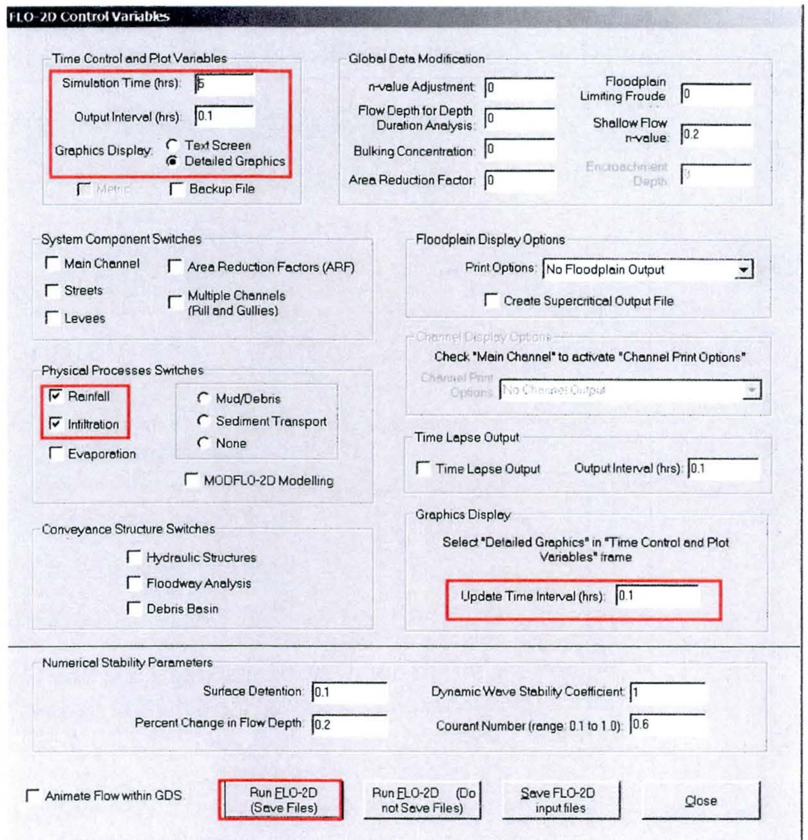


Step 16: Assign a SCS curve number from a multiple shape files

Click *Grid/Compute SCS Curve Number from Multiple Shape Files*. Select the shape files and fields as shown below and click *OK*.



Turn off the infiltration view by clicking *View/Components/Infiltration*. Click *File/RunFLO-2D...* Assign the *FLO-2D Control Variables* as shown in the dialog box and click *Run FLO-2D (Save Files)*.



Summary

This lesson demonstrated that a watershed rainfall runoff and infiltration model can be quickly developed with the GDS. The simplicity of this process enables FLO-2D to be a valuable tool for watershed and urban rainfall-runoff simulations. A more detailed storm analysis can be performed with the spatially variable rainfall tool, moving storm or NEXRAD real time rainfall simulation.

Review Questions

1. How many rainfall distributions are available as automatic input in FLO-2D?

   ____

2. True or False? It is necessary to use the rainfall distributions provided by FLO-2D?
3. Which infiltration configuration can be used with FLO-2D? Check all that apply.
   - ☐ Green-Ampt
   - ☐ SCS Curve
   - ☐ Combination of Green-Ampt and SCS Curve
4. If a combination of Green-Ampt and SCS curve infiltration is used, which method takes precedence? ____________________________________
5. True or False? Green-Ampt infiltration method removes the volume from the rainfall?
6. Two *.tbl files are needed to compute Green-Ampt infiltration data. List the data categories recorded in the two files.

   ____
   ____
   ____
   ____

7. Briefly describe when to use the 3 variations of infiltration.

   ____
   ____
   ____
   ____

8. Rainfall and infiltration help documentation is found in which locations? Check all that apply.
   - ☐ GDS Manual
   - ☐ Data Input Manual
   - ☐ Reference Manual
   - ☐ PowerPoint Presentation
   - ☐ Pocket Guide
   - ☐ Workshop Lesson Book
9. List 3 ways to assign SCS Curve infiltration in the GDS.

   ____
   ____
   ____

LESSON 8 – HEC-RAS TO FLO-2D CONVERSION

Overview

Many FLO-2D projects require improving existing flood maps created from a HEC-RAS model by using an integrated channel and floodplain flood routing analysis. This lesson will demonstrate how to load a HEC-RAS channel into the GDS and convert it into a FLO-2D channel. The HEC-RAS data provided with this lesson was edited to a shorter river reach. The channel was trimmed to just 15 cross sections and each cross section was trimmed to the top-of-bank stations. It may be necessary to edit cross section and channel data in HEC-RAS prior to importing the data to FLO-2D.

Required Data

The FLO-2D grid system files and a Geo-referenced HEC-RAS project file are needed for this lesson.

File	Content	Location*
BoiseCrk.prj	HEC-RAS Project	*Example Projects\Lesson 8*
*6 *.DAT files*	FLO-2D data files	

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To create a FLO-2D channel from HEC-RAS data using the GDS, follow these steps:

- Open the GDS program and load Lesson 8;
- Import the HEC-RAS channel;
- Create a FLO-2D channel segment;
- Load aerial image;
- Adjust the channel;

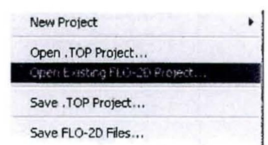
- Assign the cross section data to the channel;
- Interpolate the cross sections;
- Calculate the right bank elements;
- Realign the right bank elements;
- Create a channel outflow node;
- Save and run the model.

Step 1: Open the Grid Developer System and load Lesson 8

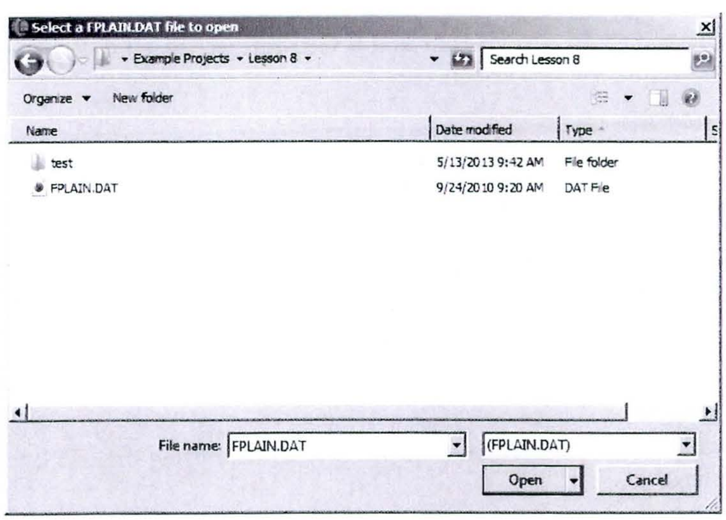
NOTE Using an Explorer window, make a new folder in the Lesson 8 folder named "TEST". Copy all the Lesson 8 files to this folder and perform the lesson in the TEST folder. ***NOTE***

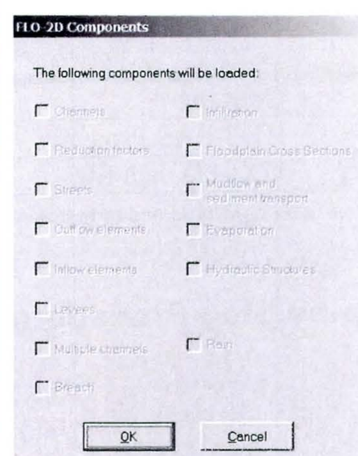
Double click the GDS Basic icon from the desktop. 

From the GDS *File* menu click *Open Existing FLO-2D Project…*



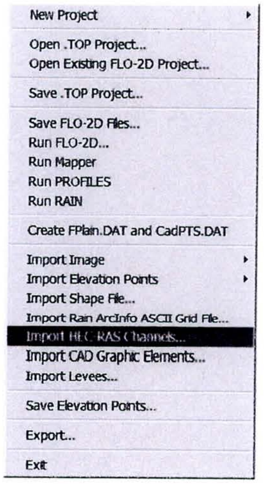
Locate the folder *Lesson 8* and open the *FPLAIN.DAT* file. Click *Open* and *OK* to load the project.



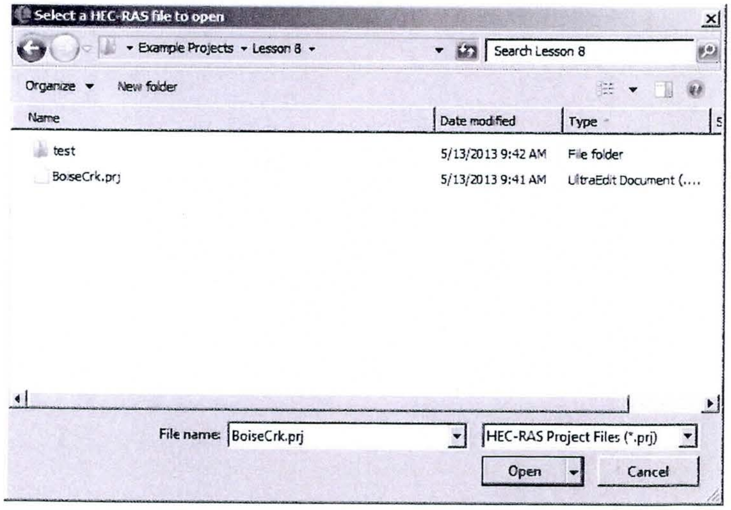


Step 2: Import the HEC-RAS channel

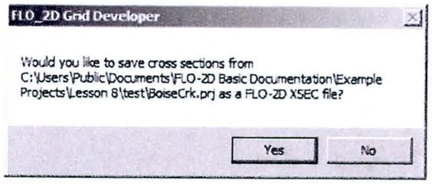
Click *File / Import HEC-RAS channels…*



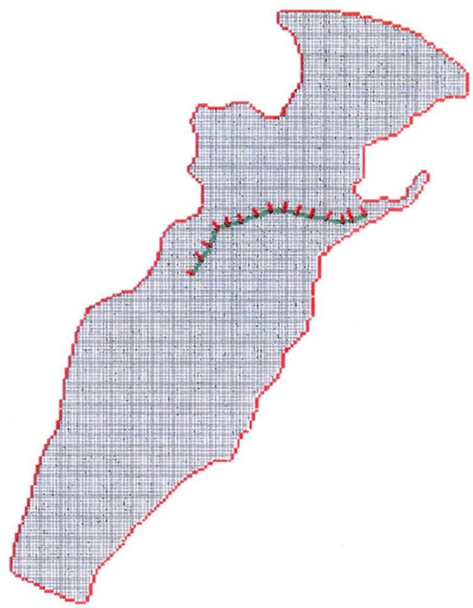
In the *Lesson 8* folder, open the *BoiseCrk.prj* file.



The GDS will automatically convert the HEC-RAS cross section data to into a FLO-2D cross section file *XSEC.DAT*. Click *Yes* and *Save* to complete the cross section conversion.

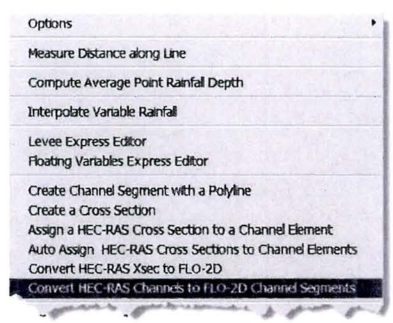


The left bank line of the HEC-RAS channel and the cross lines will be loaded in the GDS. These are reference lines to be used to create the channel features. They do not represent FLO-2D channel data. That data will be created in the remainder of the lesson.

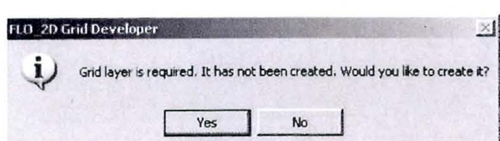


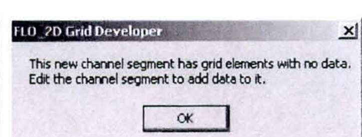
Step 3: Convert the HEC-RAS channel to a FLO-2D channel

On the *Tools* menu click *Convert HEC-RAS Channels to FLO-2D Channel Segments*.



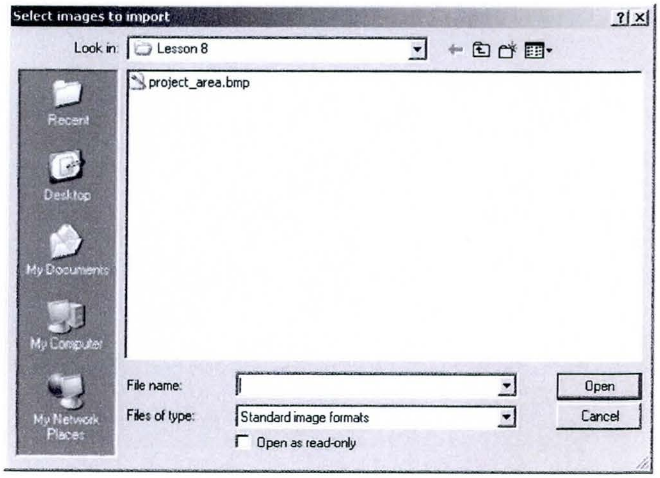
Click *Yes* to create a grid layer and *OK* to close the dialog box.





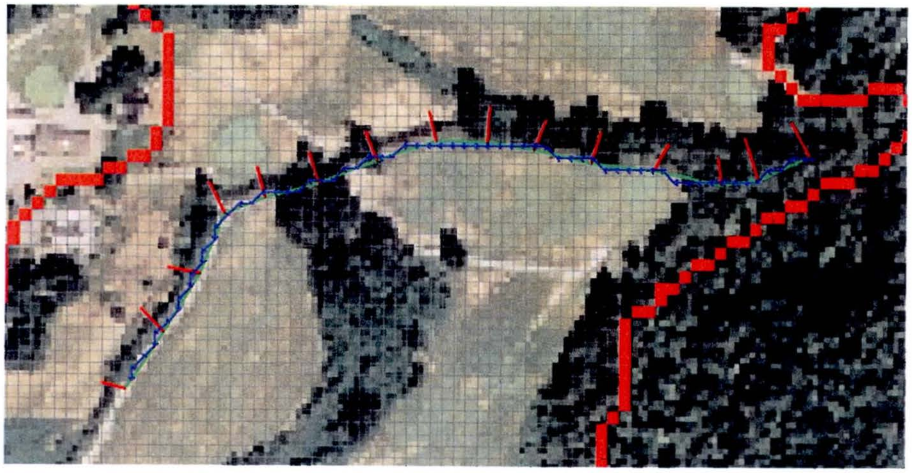
Step 4: Load aerial image

Click *File / Import Image / Individual Image…* In the *Lesson 8* folder open the **.bmp* image.

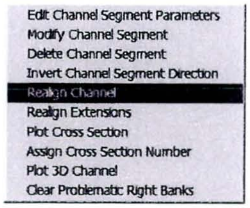


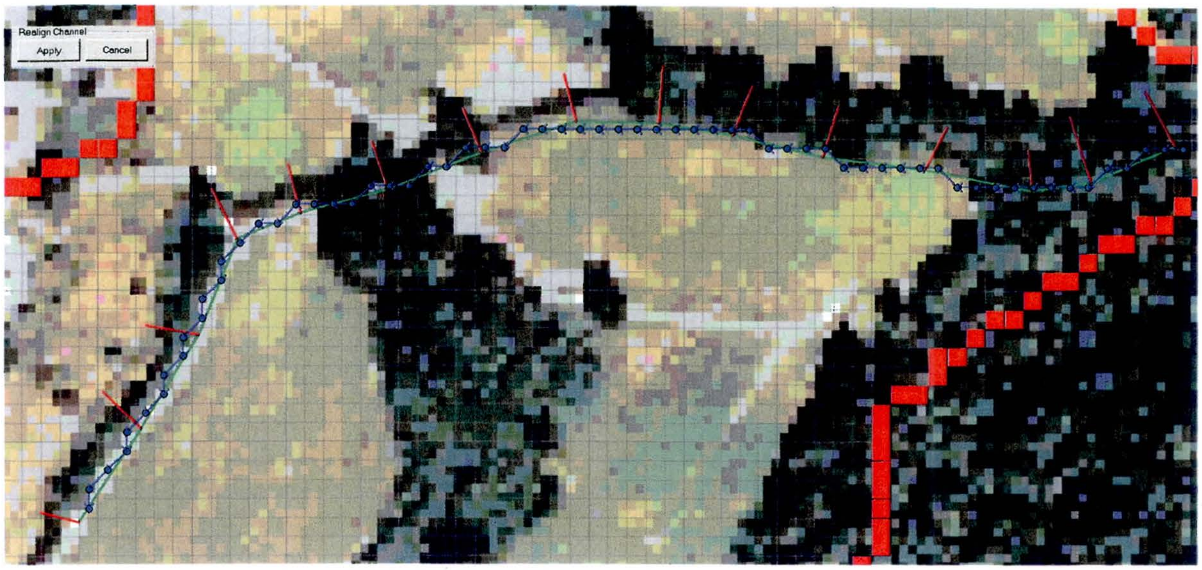
Step 5: Realign the channel segment

Zoom in on the channel.



The image below shows that the channel segment does not always align to the left bank (Green Line). It is necessary to realign the channel segment to the left bank. Left click anywhere along the channel and click *Realign Channel*.

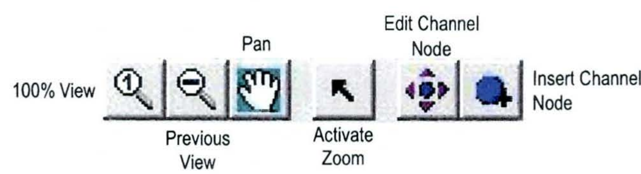




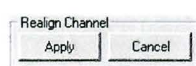
Realign the remaining channel elements using the background image and Green Line to help determine the left bank.

The align channel tools are shown below. After using pan or zoom out, start editing again by clicking the *Edit Channel* button. Click the *insert channel node* button to add a channel element.

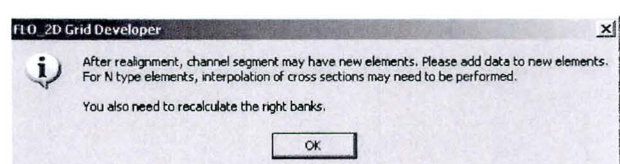
Note: To remove a channel node, drag it into a neighboring node.



To realign the channel, click and drag the blue dots to the desired grid elements. Once the channel alignment is finished, click the *Apply* button.



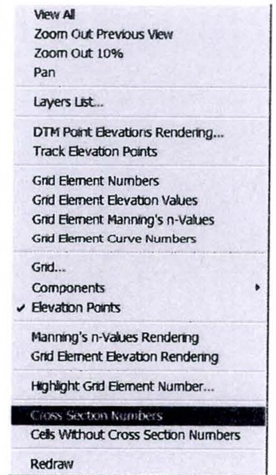
The following dialog box will appear as a reminder to add new channel element data such as channel (e.g. width and depth for rectangular geometry or cross section data for natural channels) and to recalculate the right banks. Click *OK*.



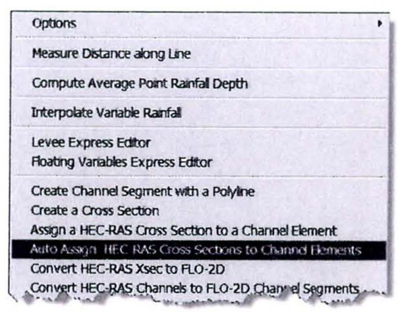
The interpolation and the right bank assignment will be completed later in this lesson.

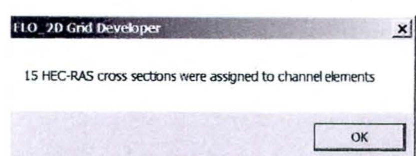
Step 6: Assign the cross section data to the channel

Save the FLO-2D data files by clicking *File/Save FLO-2D Files*. Click *View/Cross Section Numbers*.

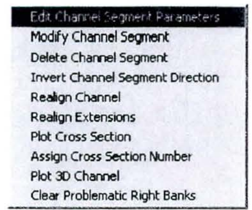


From the *Tools* menu, select *Auto Assign HEC-RAS cross sections to Channel Elements* and click *OK*.

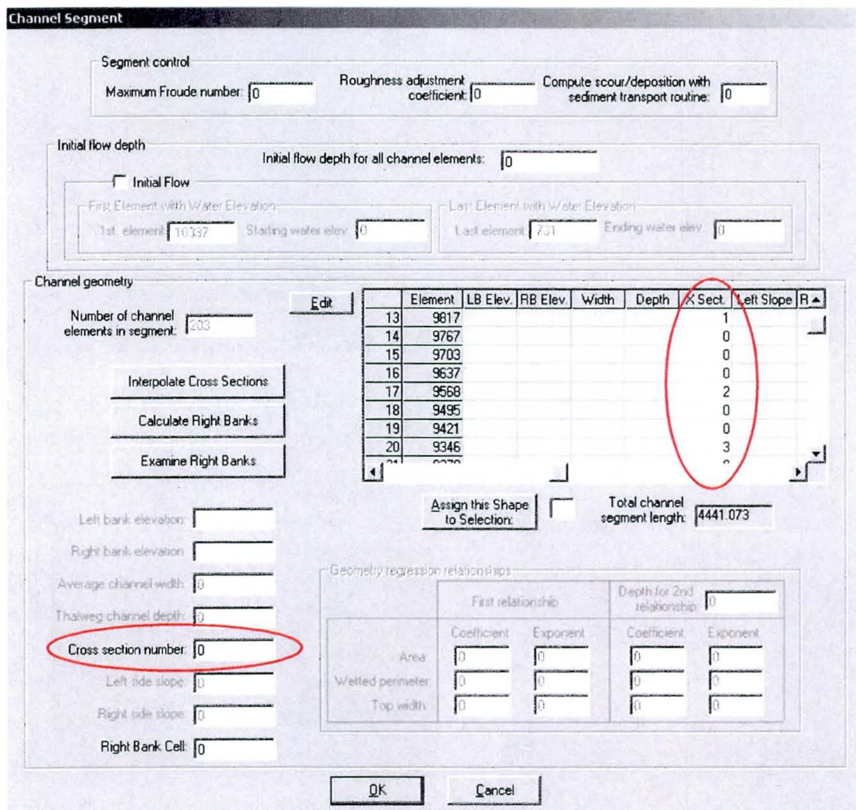




Once all HEC-RAS cross sections have been assigned click a channel element and click *Edit Channel Segment Parameters.*

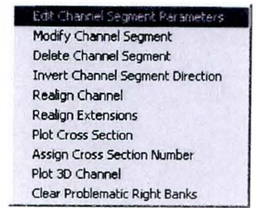


Review the list of cross sections for mistakes such as assigning the same cross section twice. Fix any potential cross section problems by changing the *Cross section number*. Click *OK* and *Yes to Validate* the table.

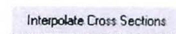


Step 7: Interpolate the cross sections

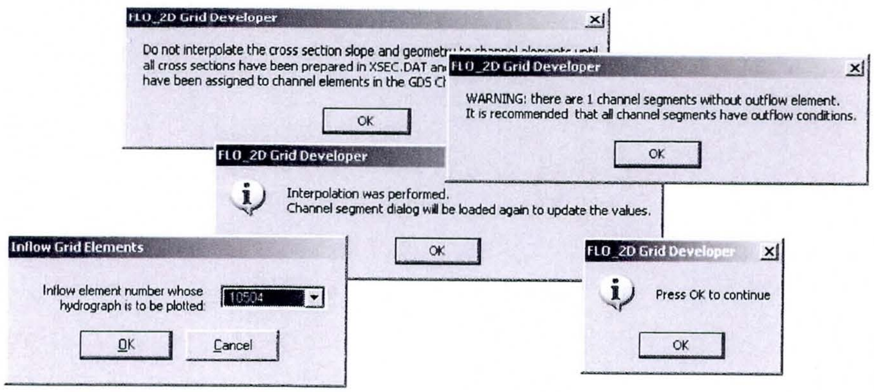
Only 15 channel elements have cross sections assigned so far. It is necessary to interpolate these cross sections to the intermediate channel elements without cross sections to complete the channel geometry. Left click anywhere on the channel and click *Edit Channel Segment Parameters*.



Click the *Interpolate Cross Sections* button on the *Channel Segment* dialog box.

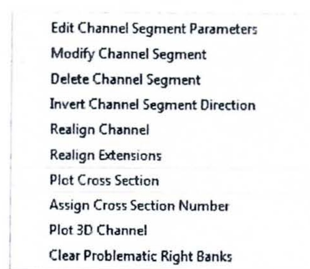


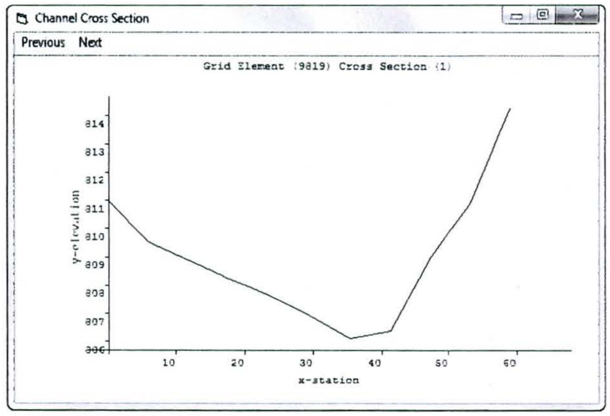
A series of message boxes will appear. Click *OK* through all of them.



Now each channel element has an assigned or interpolated cross section and the XSEC.DAT file has been updated. Note that the original cross section names are now preceded by X- in the XSEC.DAT file.

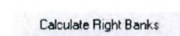
Once all the cross sections have been interpolated, a cross section plot for each channel element can be viewed. Left click the channel and click *Plot Cross Section*. On the cross section plot, the *Previous* and *Next* button options can be used to move upstream and downstream through the channel elements.

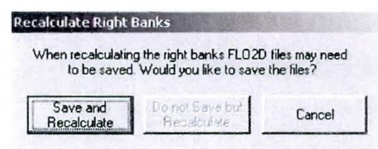




Step 8: Calculate the right bank elements

Click the *Calculate Right Banks* button on the *Channel Segment* dialog box. Click *Save and Recalculate.*

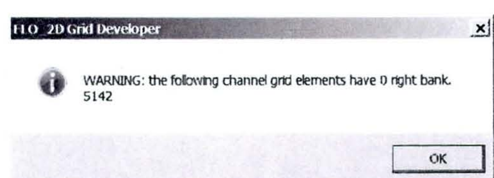




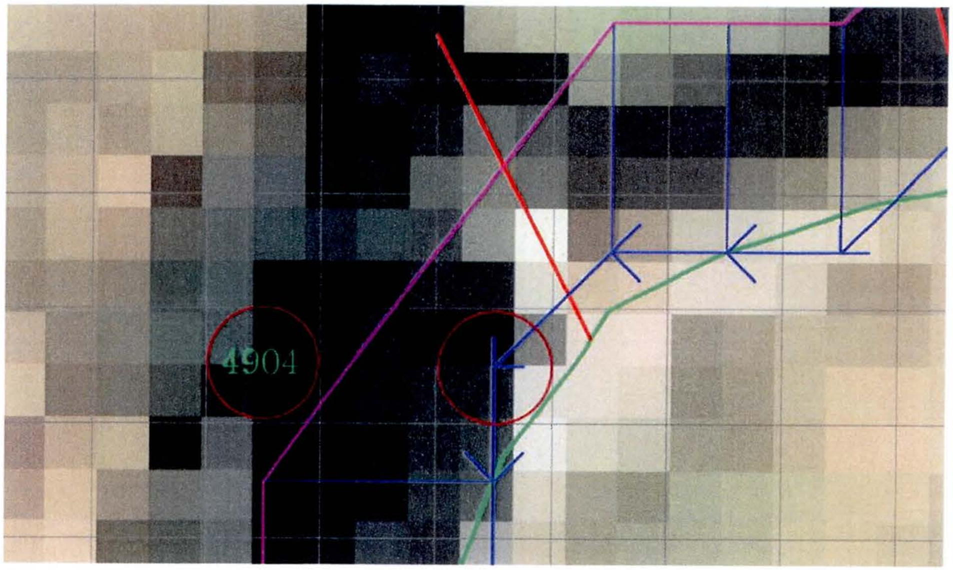
Click *OK* through the series of dialog message boxes. Note the last column of channel right banks in the *Channel Segment* dialog box is now filled.

Step 9: Fill in missing right bank elements and realign right banks

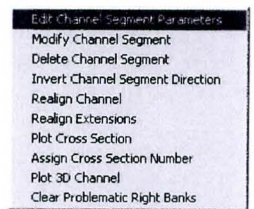
It may now be necessary to manually adjust the right bank elements. In some cases, the right bank element is not calculated an must be input manually.



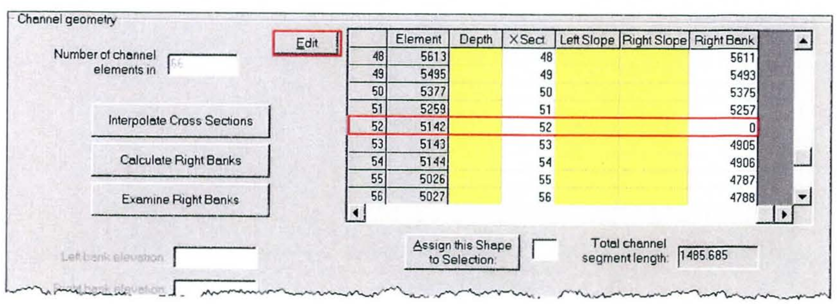
Zoom in on a missing right bank. Choose a right bank element that follows the general line of the right bank. *Write down* the grid element number of the desired right bank element.



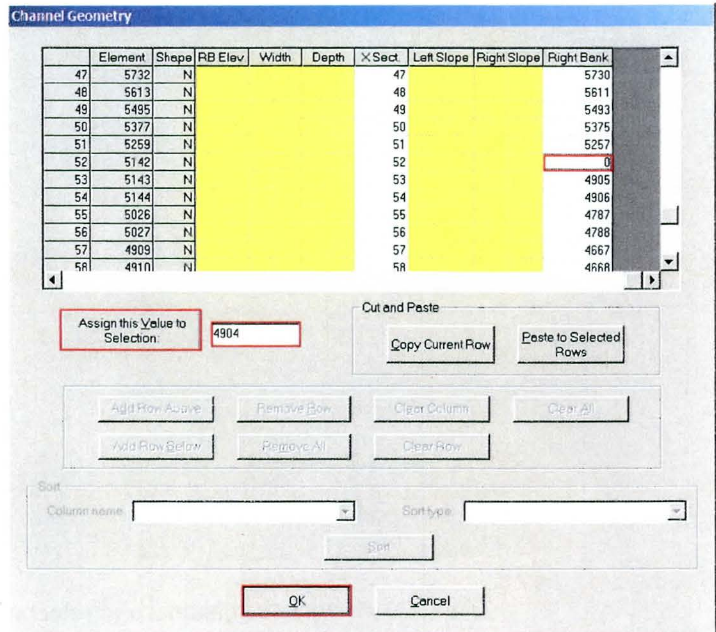
Left click the channel and select *Edit Channel Segment Parameters*.



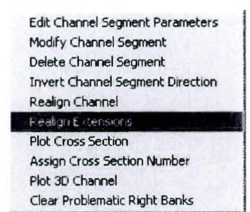
On the channel geometry dialog box, click the *Edit* button.

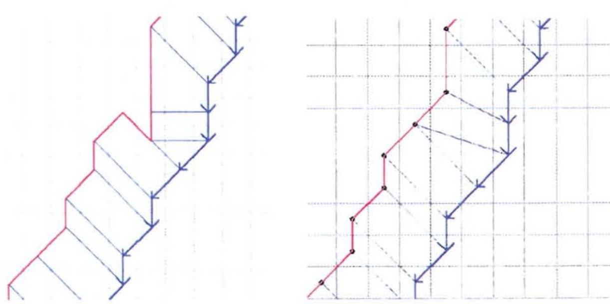


Select the missing right bank element by clicking on the cell with a *zero* in it. Enter the desired right bank element in the *Assign this Value to Selection* dialog entry and click the button. When finished, click *OK* to close the *Channel Geometry* dialog box and *OK* to close the *Channel Segment* dialog box and *Yes* to *Validate the Channel Variables*.



Zoom in on the upstream section of the channel and pan downstream until a right bank adjustment is desired. Click on the channel and click *Realign Extensions*.



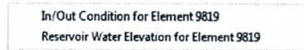


Once the right bank elements are aligned *save the FLO-2D* files.

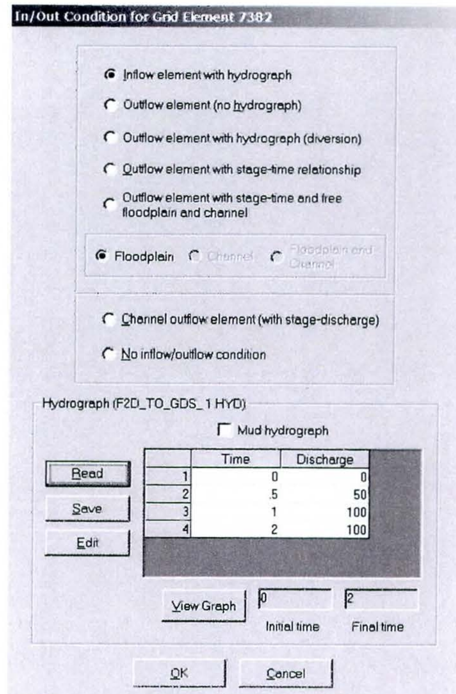
Step 10: Create a channel inflow and outflow node

Inflow Node

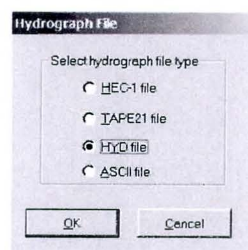
Right click on the first channel element and click *In/Out Condition for Element *****.



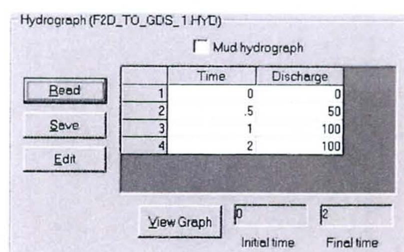
Fill out the *In/Out Condition* dialog box as shown below and click *Read*.

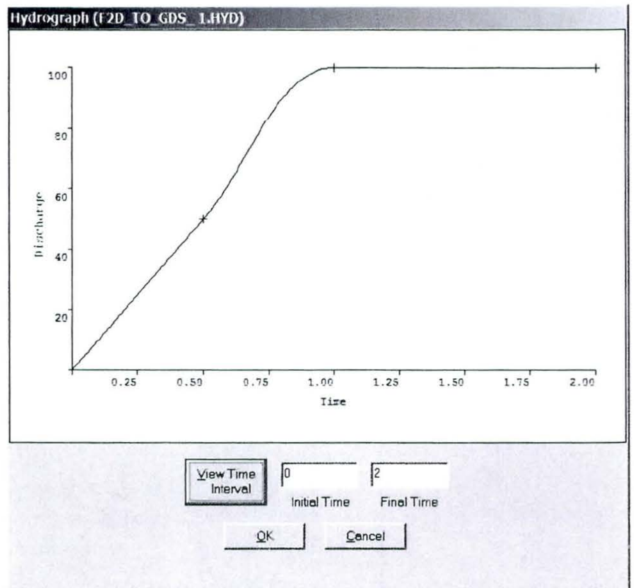


Select the *HYD file* and click *OK*.



Load the file *F2D_TO_GDS_1.HYD*. Click the *View Graph* button.



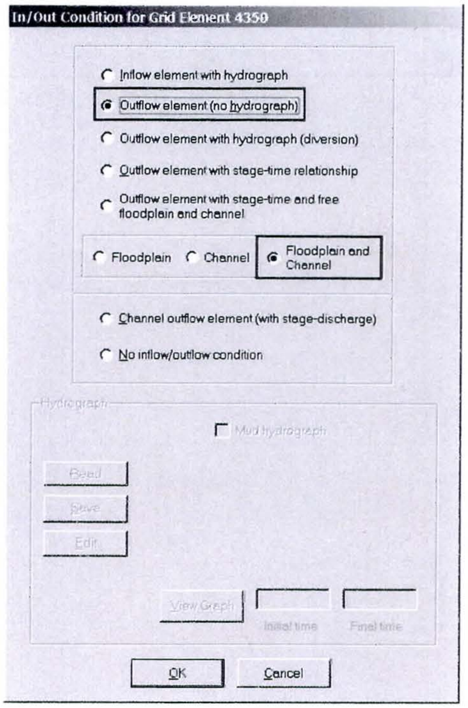


Click *OK* to close the plot and then click *OK* to accept the inflow hydrograph.

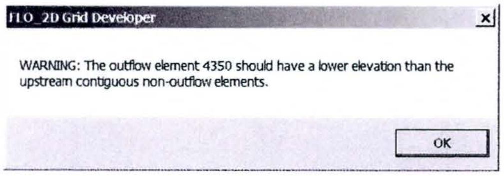
Outflow Node

Right click on the last channel element and click *In/Out Condition for Element *****.

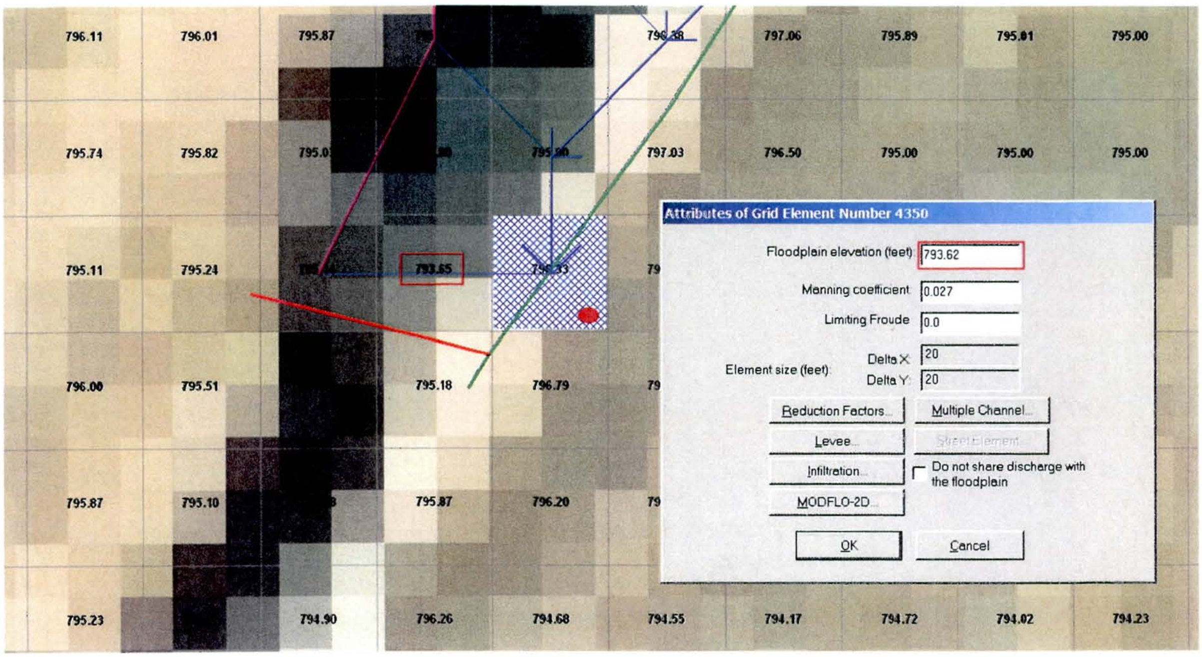
Fill out the *In/Out Condition* dialog box as shown below and click *OK*.



In this case, the outflow node is higher than one of its neighboring grid elements.



The outflow cannot be calculated correctly unless there is a negative slope. Create a negative slope by lowering the grid element elevation at the outflow node to just less than the lowest neighboring grid element. In this case the lowest elevation is *793.65*. *Double click* the outflow grid element near the red oval to activate the *Attributes of Grid Element Number *****. Lower the elevation and select *OK*.



Inflow and outflow parameters are defined and the model is ready to run.

Step 11: Save and Run

Click *File/Run FLO-2D…* Make sure the *FLO-2D Control Variables* dialog box is filled in like the one below. Click *Run FLO-2D (Save Files)*.

Summary

This GDS lesson demonstrates the steps required to create a natural channel from a HEC-RAS channel. The HEC-RAS data used in this lesson was edited to fit FLO-2D top-of-bank stations. All cross sections were trimmed to the bank stations and the channel was trimmed to just 15 cross sections. The lesson also illustrates the channel editing tools available in the Basic GDS.

Review Questions

1. Identify three types of channels that can be used in FLO-2D?

 ____

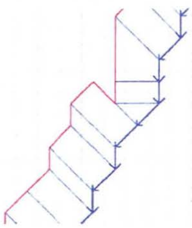
 ____

 ____

2. True or False It is necessary to have a Geo-referenced HEC-RAS project to import channel into the GDS?
3. Which bank is the channel segment assigned to?
 - ☐ Right bank looking downstream
 - ☐ Left bank looking downstream
 - ☐ Channel thalweg
4. Which tool is used to convert HEC-RAS cross section data when the project is not geo-referenced? ______________________
5. True or False It is recommended to align the left bank prior to entering the channel segment attribute data?
6. Which program is used to view and edit channel bed slope and cross section data?

 ____

7. The right bank assignment in the following image is incorrect. Explain how to fix it.



 ____

 ____

 ____

 ____

 ____

 ____

8. If a channel outflow node is assigned to the last channel element of a segment and the slope is negative the model will terminate with an error. Explain how to fix this problem.

 ____

 ____

 ____

9. Channel help documentation is found in which locations? Check all that apply.
 - ☐ License Documentation
 - ☐ Data Input Manual
 - ☐ Reference Manual
 - ☐ PowerPoint Presentation
 - ☐ Pocket Guide
 - ☐ Workshop Lesson Book

LESSON 9 – HYDRAULIC STRUCTURES

Overview

Discharge is controlled by hydraulic structures using either rating curves or rating table data in FLO-2D. A rating curve or table can be made for bridges, weirs, culverts and other hydraulic structures using HEC-RAS, weir equations, culvert equations/tables or various hydraulic structure software programs. Hydraulic structure data is entered using the GDS. This lesson details the process of creating hydraulic structure data files.

Required Data

The FLO-2D grid system files and a hydraulic structure rating table data file are needed for this lesson.

File	Content	Location*
*6 *.DAT files*	FLO-2D data files	*\Example Projects\Lesson 9*
Rating table...	Rating table data	*\Example Projects\Lesson 9*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To create a FLO-2D channel from HEC-RAS data using the GDS, follow these steps:

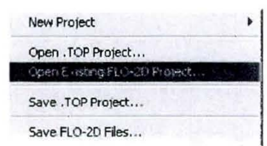
- Open the GDS program and load Lesson 9;
- Load aerial image;
- Locate and create all hydraulic structures;
- Enter rating table data for the detention basin outlet works;
- Save the project;
- Manually enter the rating table data;
- Reload project in GDS and run the model.

Step 1: Open the Grid Developer System and load Lesson 9

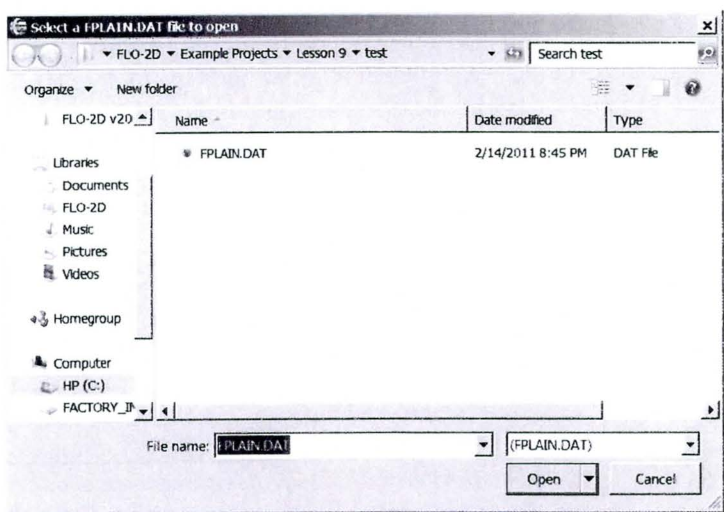
NOTE Using an Explorer window, make a new folder in the Lesson 9 folder named "TEST". Copy all the lesson 9 files to this folder and perform the lesson in the TEST folder. ***NOTE***

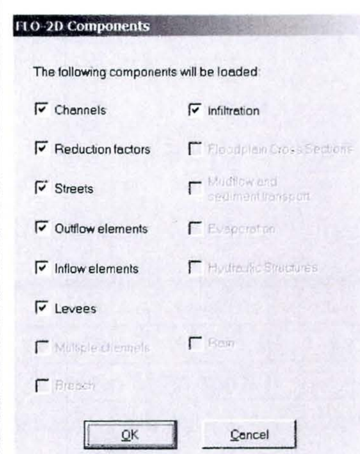
Double click the GDS Basic icon from the desktop.

From the GDS *File* menu click *Open Existing FLO-2D Project...*



Locate the folder *Lesson 9/TEST* and open the *FPLAIN.DAT* file. Click *Open* and *OK* to load the project.

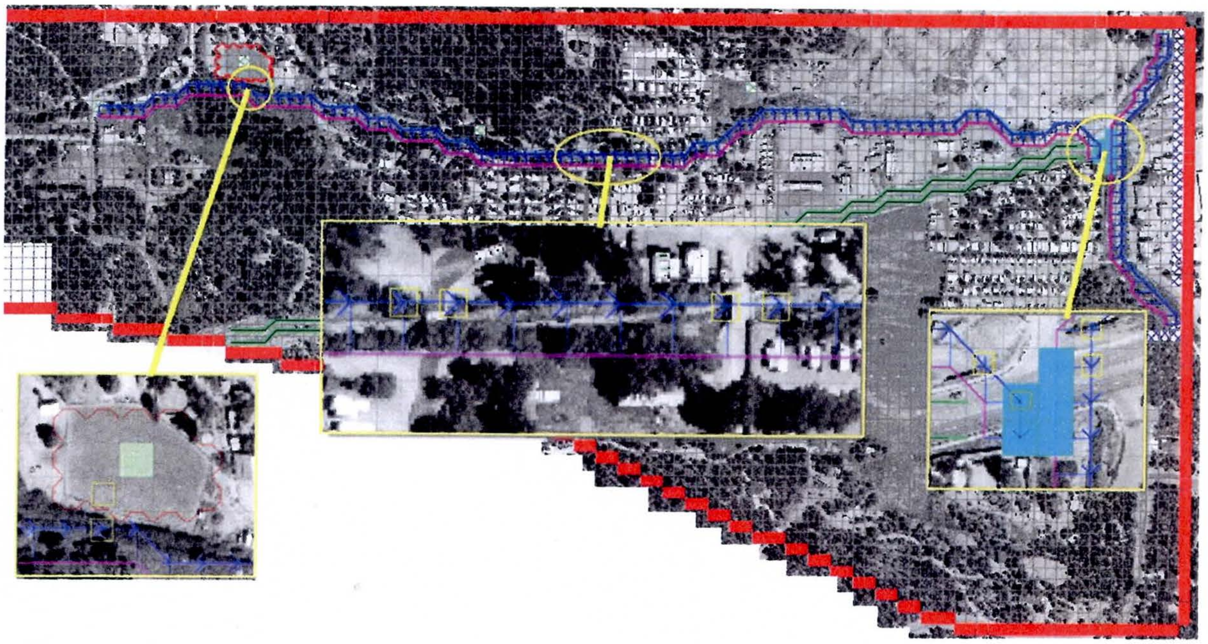




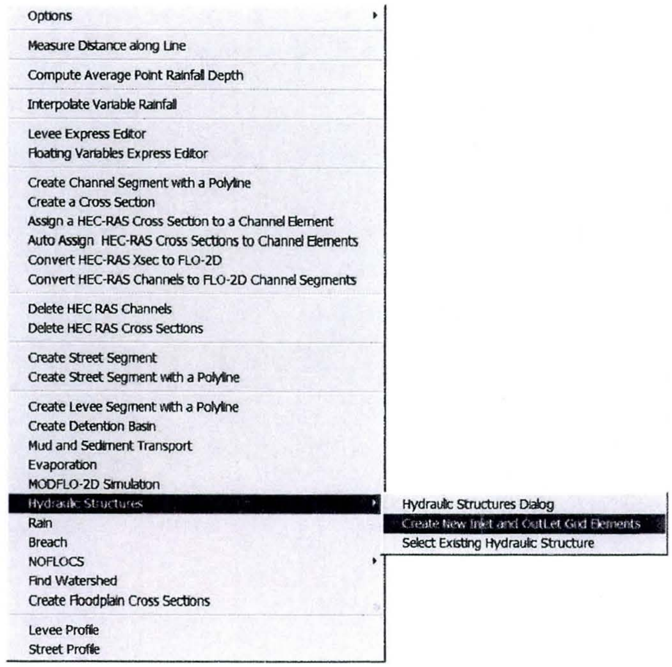
Step 2: Importing aerial images (see Lesson 1 - Step 4)

Step 3: Locate and create all five hydraulic structures

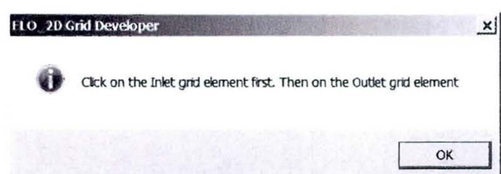
Use the following image to locate and enter 5 hydraulic structures.

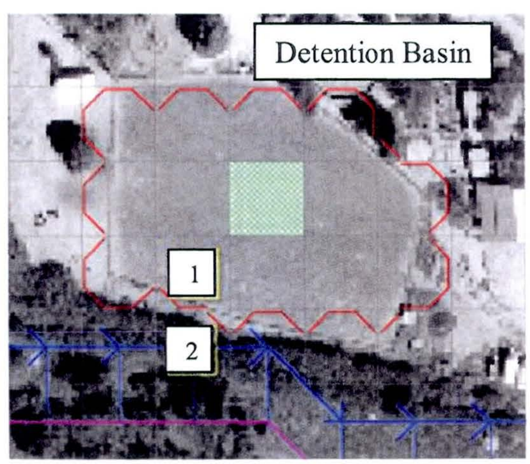


The first hydraulic structure is the outlet works for a detention basin. Click *Tools/ Hydraulic Structures/Create New Inlet and Outlet Grid Element*.



Click *OK* then click the two grid elements in the order shown in the following image:

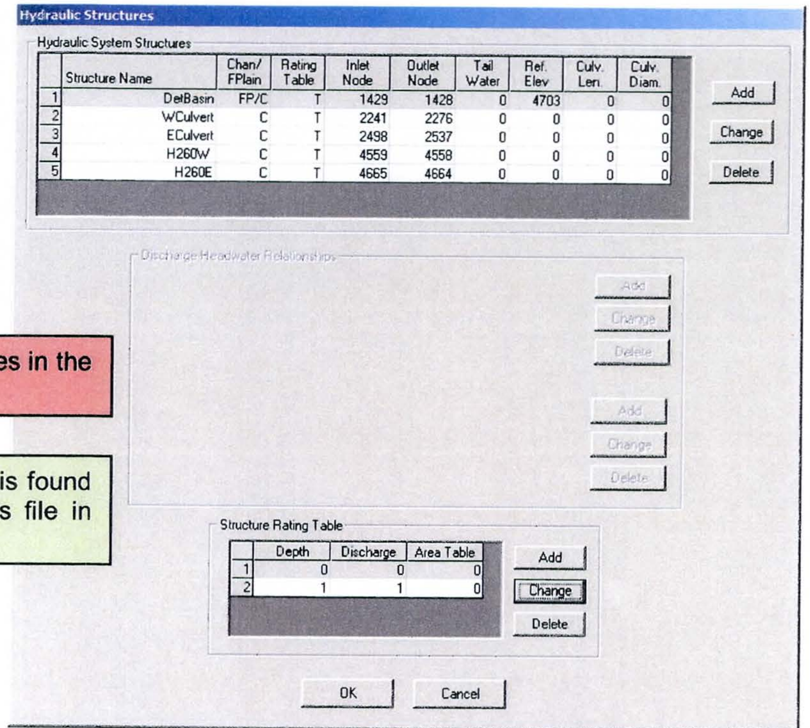




Once the outlet element is clicked, the *Hydraulic Structures* dialog box will appear. The *Change* button in the *Hydraulic System Structures* group allows the user to enter the data as shown in the *Add Hydraulic Structure* dialog box. All structures are presented in the following image. Use it for reference.

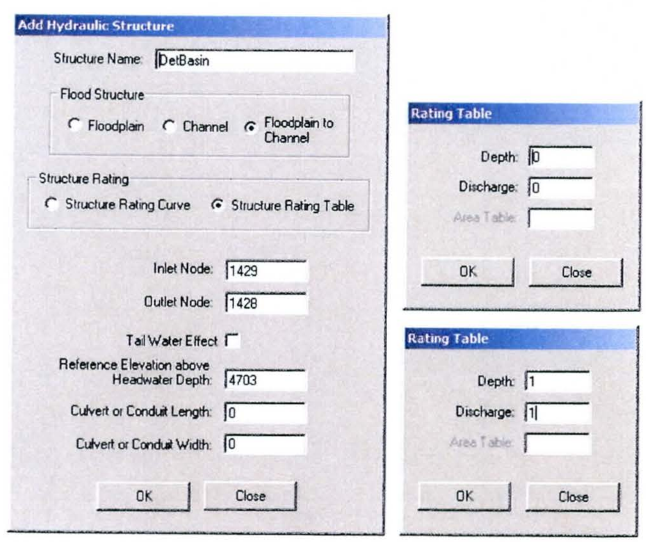
Warning: Do not enter spaces in the Name field.

Hint: The rating table data is found in the Rating Table Data.xls file in the Lesson 9 folder.

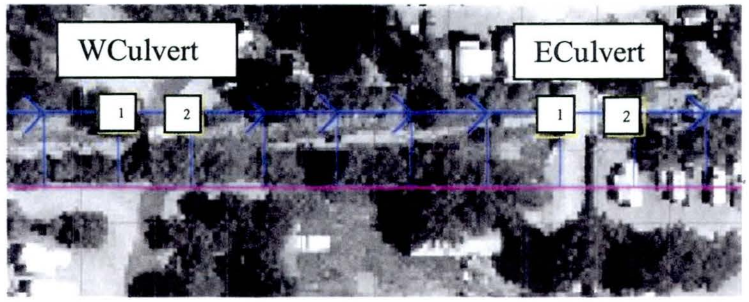


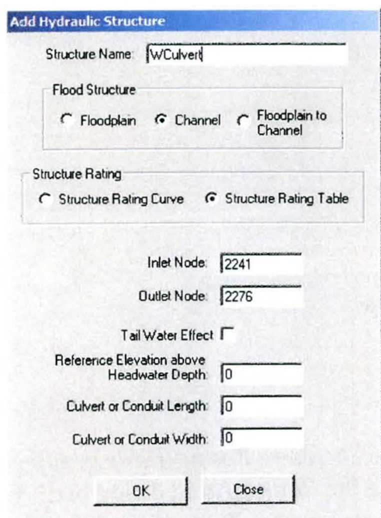
The *Change* button in the *Structure Rating Table* group is used to enter the rating table data using the *Rating Table* dialog box. Enter a rating table of 0,0 and 1,1 for all structures. Data will be filled in for all tables at once later.

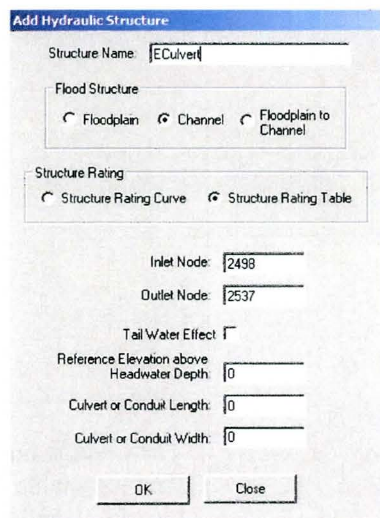
Hint: The Headwater Depth is only used when the structure doesn't turn on until the water surface elevation must reach a specified value.



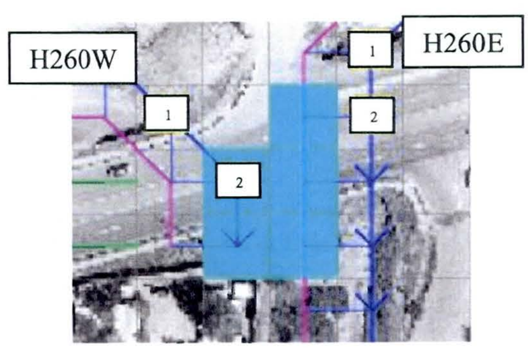
Zoom in on the second group of hydraulic structures and assign the inlet and outlet node for each structure. Click *Tools/ Hydraulic Structures/Create New Inlet and Outlet Grid Element*. Enter the Hydraulic Structure control data and 0,0 1,1 rating table data. The rating table data will be filled for all structures in Step 5.

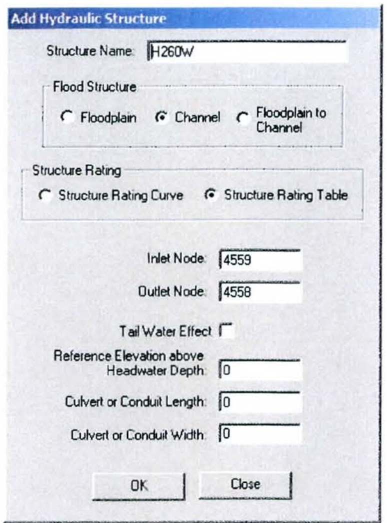


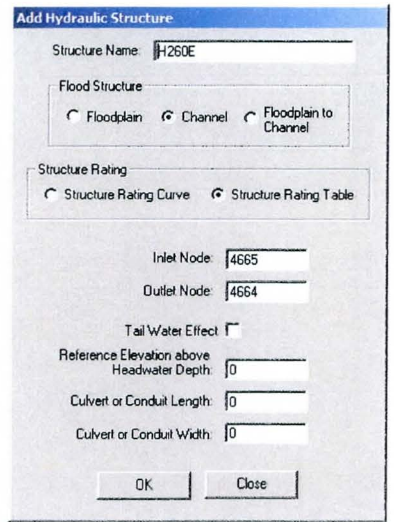




Zoom in on the final group of hydraulic structures and assign the inlet and outlet node for each. Click *Tools/ Hydraulic Structures/Create New Inlet and Outlet Grid Element*. Enter the Hydraulic Structure control data but do not enter the rating table data.

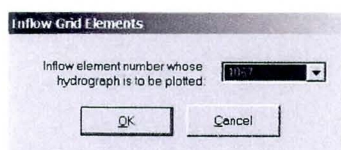


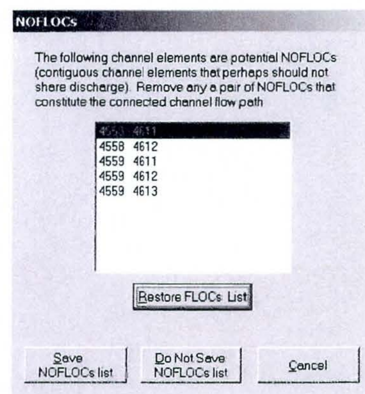




Step 4: Save the FLO-2D data files.

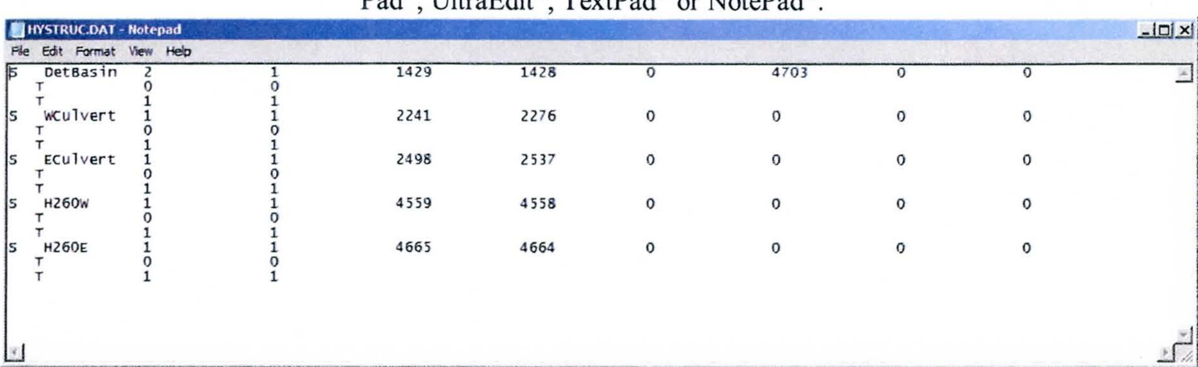
Click *File/Save the FLO-2D Data Files* to the *Lesson 9 Test* folder. Click *OK* to select an inflow hydrograph for the graphic display. Click *Save NOFLOCs List*.





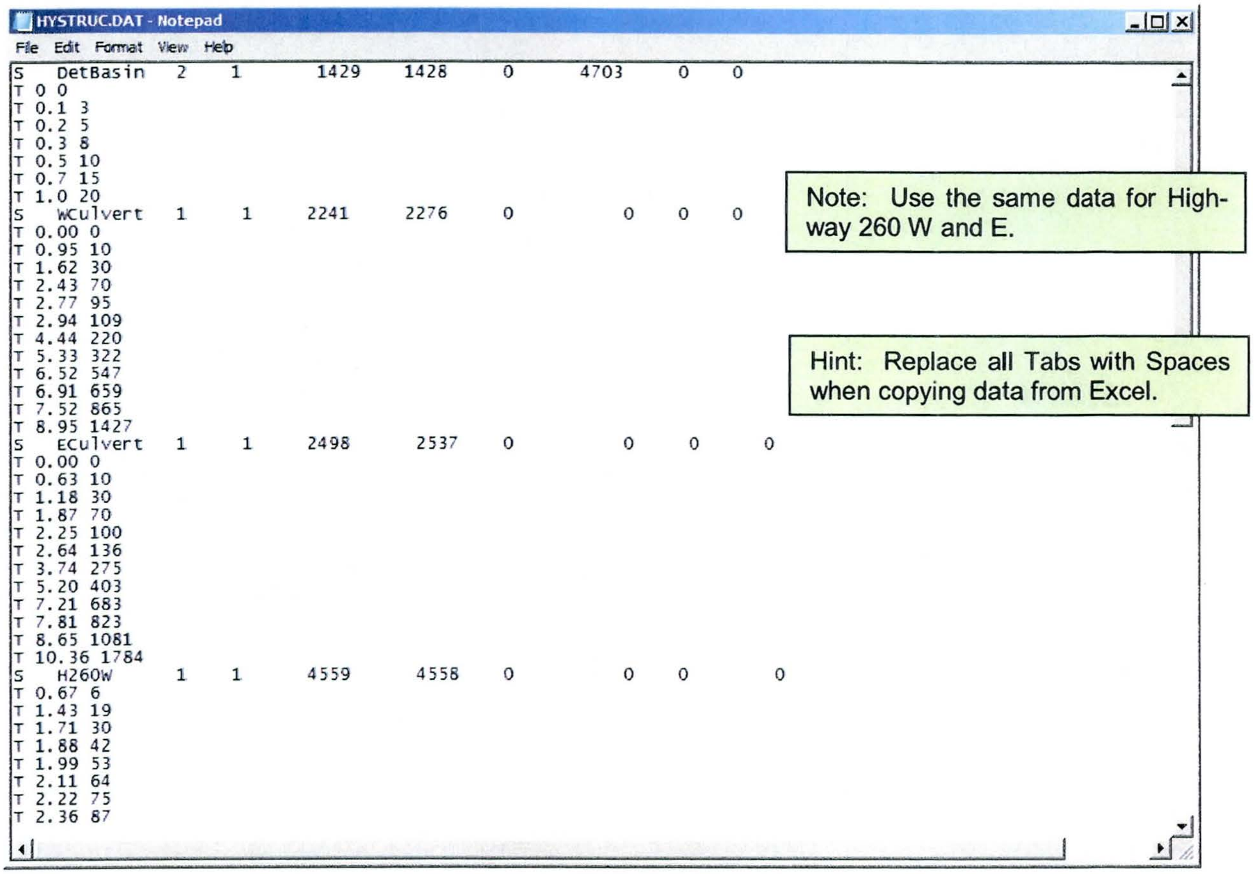
Step 5: Fill out the hydraulic structure rating table data

Open the *HYSTRUC.DAT* file in the Lesson 9/Test folder using WordPad©, UltraEdit©, TextPad© or NotePad©.




```
S   DetBasin   2   1   1429   1428   0   4703   0   0
 T             0   0
 T             1   1
S   WCulvert   1   1   2241   2276   0   0   0   0
 T             0   0
 T             1   1
S   ECulvert   1   1   2498   2537   0   0   0   0
 T             0   0
 T             1   1
S   H260W      1   1   4559   4558   0   0   0   0
 T             0   0
 T             1   1
S   H260E      1   1   4665   4664   0   0   0   0
 T             0   0
 T             1   1
```

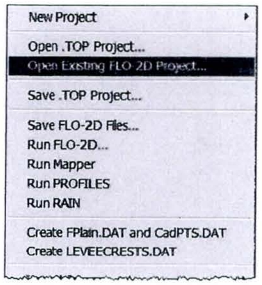
Cut and paste from the Excel file to complete the HYSTRUC.DAT file. It may be necessary to replace tabs with spaces if using NotePad or WordPad.

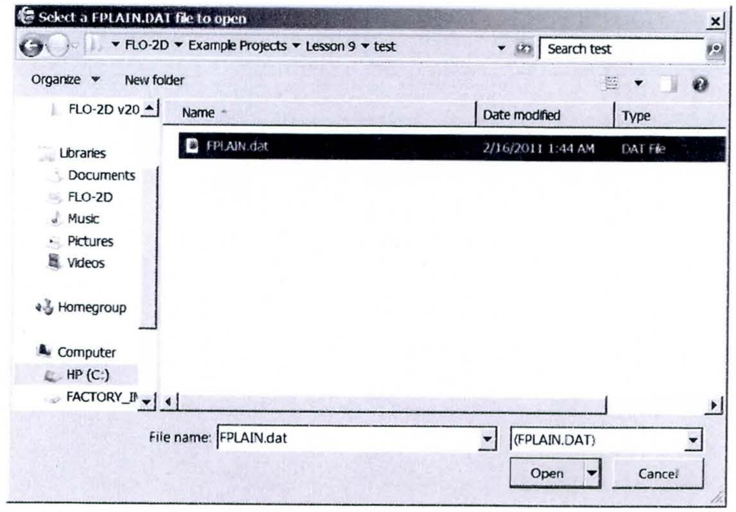


```
HYSTRUC.DAT - Notepad
File Edit Format View Help
S   DetBasin   2    1      1429     1428     0      4703    0    0
T 0 0
T 0.1 3
T 0.2 5
T 0.3 8
T 0.5 10
T 0.7 15
T 1.0 20
S   WCulvert   1    1      2241     2276     0         0    0    0
T 0.00 0
T 0.95 10
T 1.62 30
T 2.43 70
T 2.77 95
T 2.94 109
T 4.44 220
T 5.33 322
T 6.52 547
T 6.91 659
T 7.52 865
T 8.95 1427
S   ECulvert   1    1      2498     2537     0         0    0       0
T 0.00 0
T 0.63 10
T 1.18 30
T 1.87 70
T 2.25 100
T 2.64 136
T 3.74 275
T 5.20 403
T 7.21 683
T 7.81 823
T 8.65 1081
T 10.36 1784
S   H260W      1    1      4559     4558     0         0    0       0
T 0.67 6
T 1.43 19
T 1.71 30
T 1.88 42
T 1.99 53
T 2.11 64
T 2.22 75
T 2.36 87
```

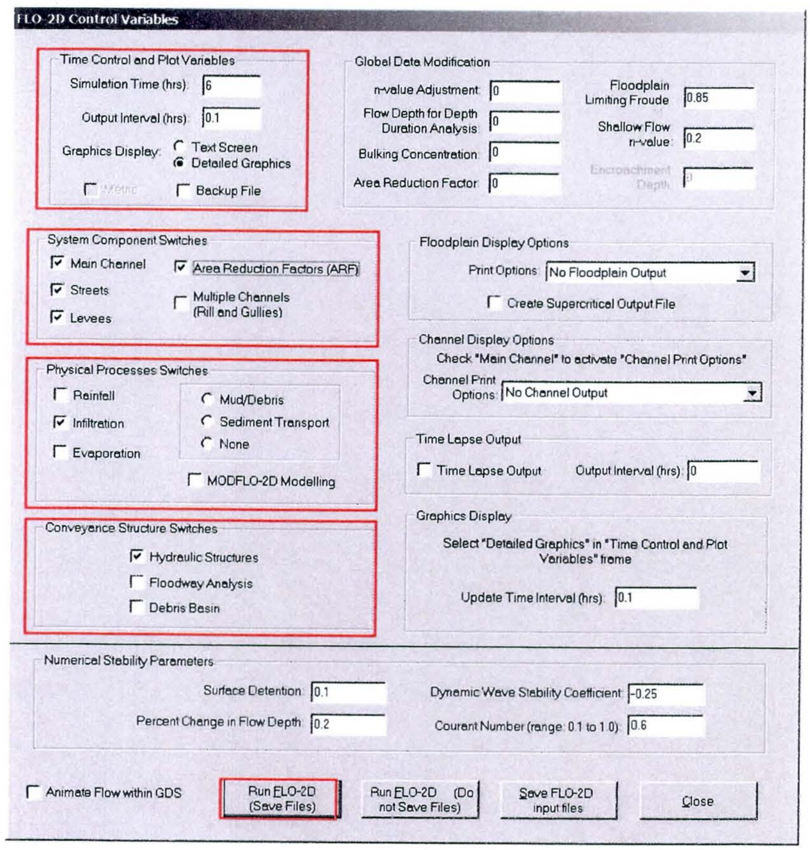

Step 6: Reload the project in GDS and run the model

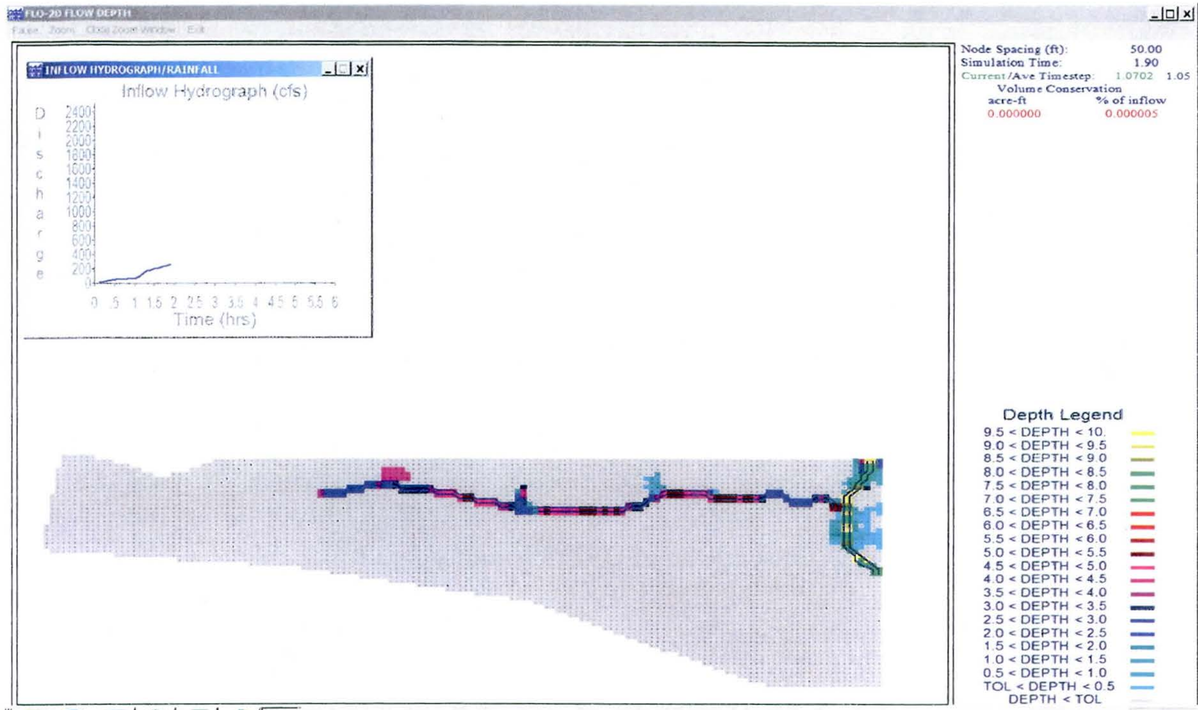
The project data files were edited outside the GDS. Reload the project by clicking *File/Open Existing FLO-2D Project...*





Click File/Run FLO-2D and fill out the Control Variables dialog box as shown below. Click Run (Save Files).





Review Questions

1. Name 3 sources of rating table data?

   ____

   ____

   ____

2. True or False? A hydraulic structure can transfer flow from a channel to the floodplain?
3. True or False? A hydraulic structure can transfer flow both upstream and downstream?
4. How many cross sections are needed to create a hydraulic structure rating table in HEC-RAS? ____________________________________
5. True or False? A channel hydraulic structure cannot have any channel elements in between the upstream and downstream nodes?
6. Which output file reports hydraulic structure flow data? ________________________
7. Name 5 different hydraulic structures that can be modeled with a rating table:

   ____

   ____

   ____

   ____

   ____

8. Briefly explain how the head reference elevation works for a hydraulic structure:

____

____

____

9. Hydraulic structure help documentation is found in which locations? Check all that apply:

- ☐ License Documentation
- ☐ Data Input Manual
- ☐ Reference Manual
- ☐ PowerPoint Presentation
- ☐ Pocket Guide
- ☐ Workshop Lesson Book

LESSON 10 – LEVEES, WALLS AND BERMS

Overview

This lesson outlines the procedure for creating and editing levee data using the GDS. The FLO-2D levee component can be used to simulate any embankment feature including berms, roadway and railroad embankments, floodwalls, sound walls, and privacy walls.

Required Data

The FLO-2D grid system files are needed for this lesson.

File	Content	Location*
*6 *.DAT files*	FLO-2D data files	*\Example Projects\Lesson 10*
Levee Points.xyz	Levee Polyline Data	*\Example Projects\Lesson 10*
LeveePolyGoat.xyz	Levee Polyline Data	*\Example Projects\Lesson 10*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To create a embankment structure using the FLO-2D levee component in the GDS, follow these steps:

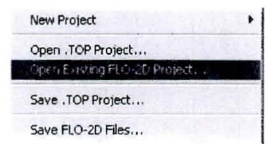
- Open the GDS program and load Lesson 10;
- Load aerial image;
- Create berm around detention basin;
- Build a levee along channel using the polyline tool;
- Create a sound wall with a polyline;
- Create a levee from an imported 3-D polyline;
- Import levee along levee crest points;
- Save and run.

Step 1: Open the Grid Developer System and load Lesson 10

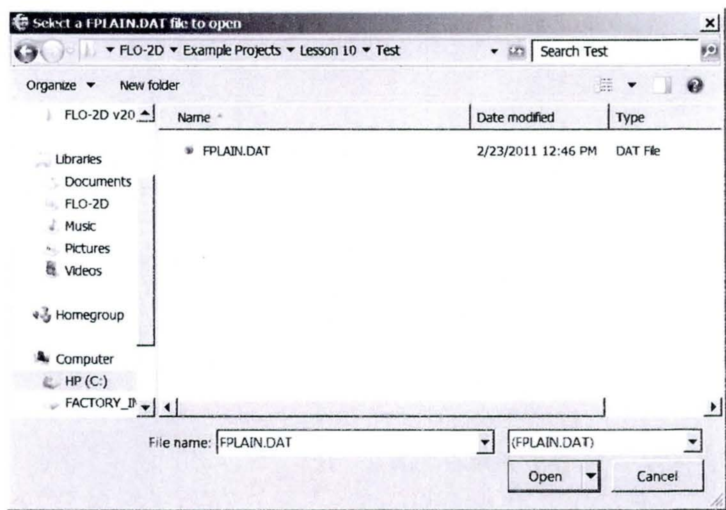
NOTE Using an Explorer window, make a new folder in the Lesson 10 folder named "TEST". Copy all the Lesson 10 files to this folder and perform the lesson in the TEST folder. ***NOTE***

Double click the GDS Basic icon from the desktop. 

From the GDS *File* menu click *Open Existing FLO-2D Project...*



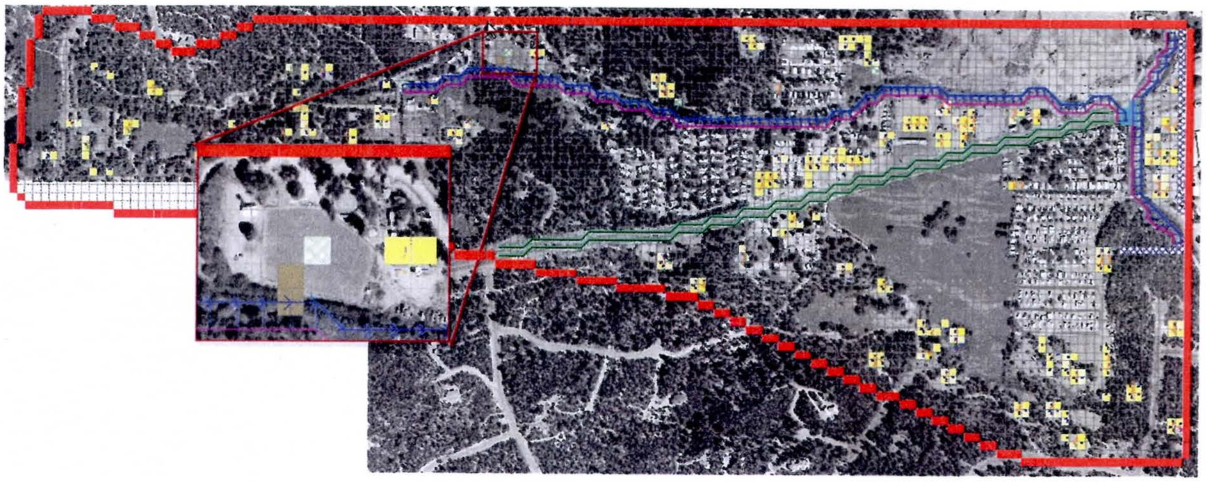
Locate the folder *Lesson 10/TEST* and open the *FPLAIN.DAT* file. Click *Open* and *OK* to load the project.



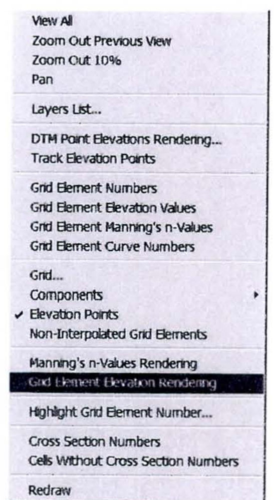
Step 2: Importing aerial images (see Lesson 1 - Step 4)

Step 3: Create a berm around the detention basin.

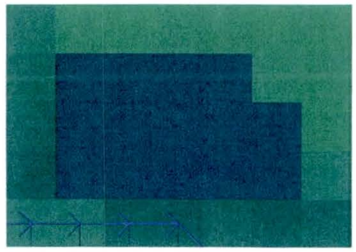
Zoom in on the detention basin. This detention basin was created by lowering the grid element elevations. The berm (using the levee component) will increase the detention basin storage.

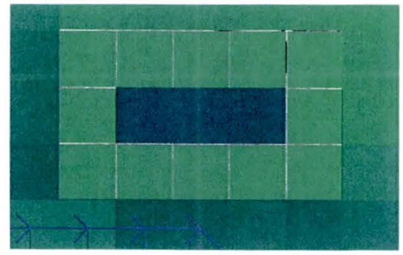


The elevation of each cell within the detention basin was lowered to 4702 ft. Click *View/Grid Element Elevation Rendering*.

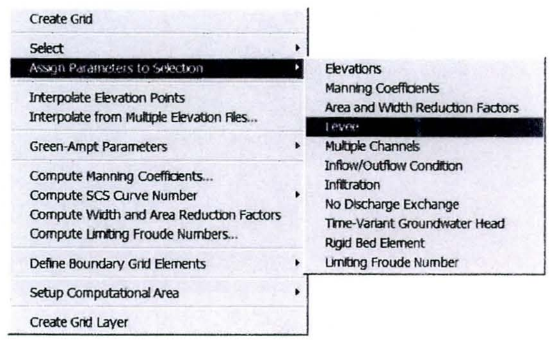


Use the *Select Element by Element* button to select the grid elements surrounding the detention basin.

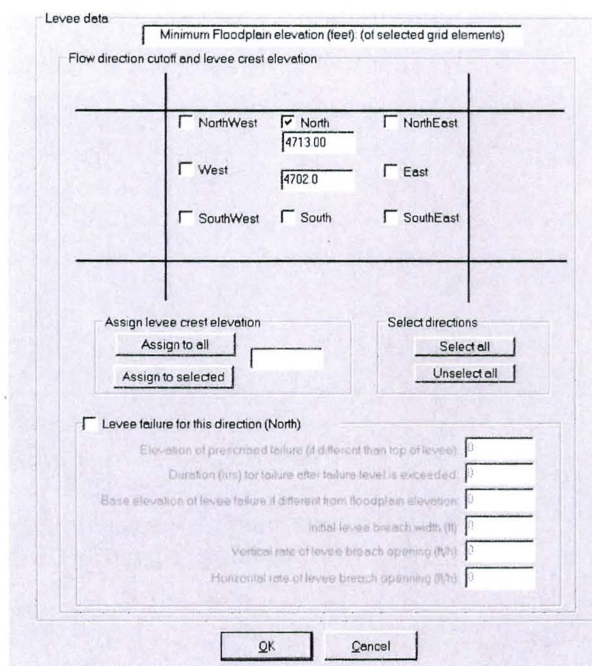




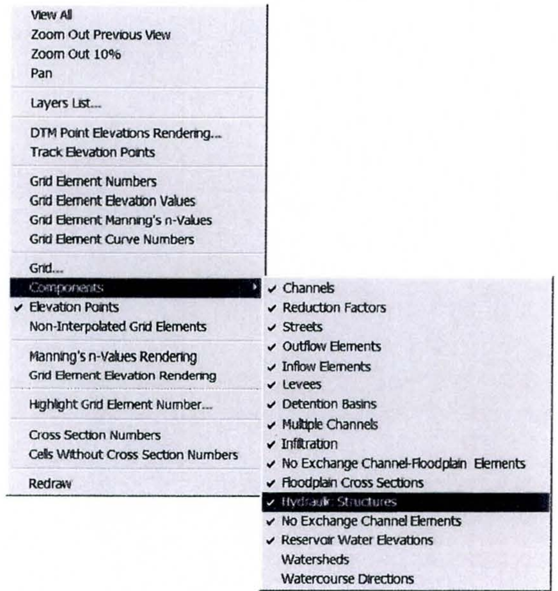
Turn off the elevation rendering by clicking *View/Grid Element Elevation Rendering*. Click *Grid/Assign Parameters to Selection/Levee.*



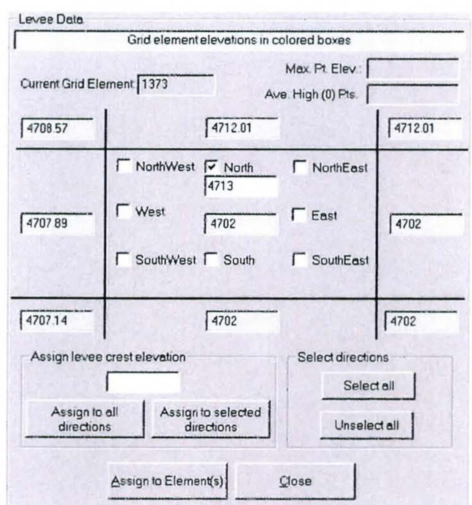
Assign a levee crest elevation of *4713.00* ft to the *north* cutoff direction and click *OK*.



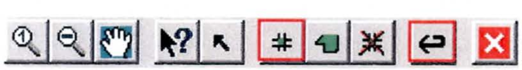
Hide the hydraulic structures by clicking *View/Components/Hydraulic Structures*.



Click the *Levee Express Editor* button and build a levee around the detention basin. Double click an element with a levee in it to activate the levee express editor.

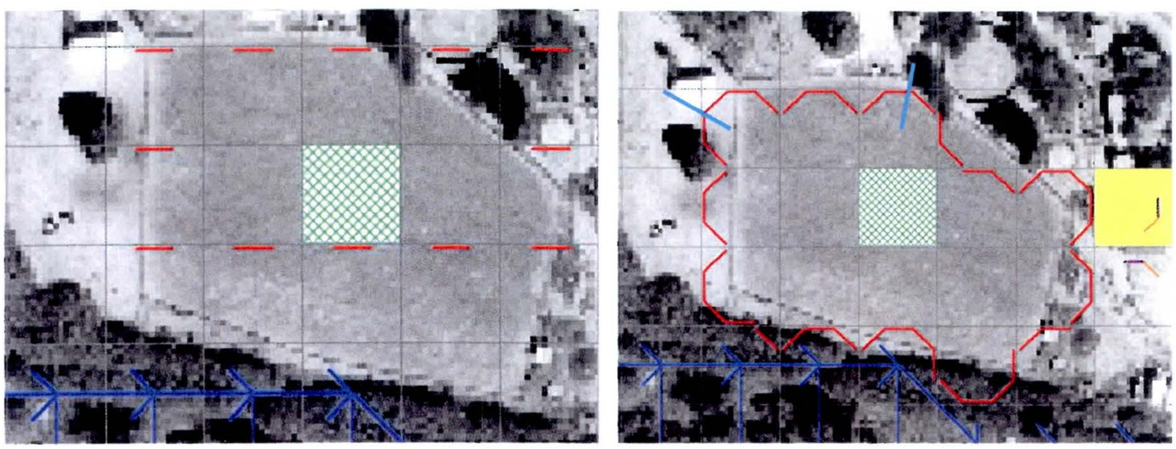


Grid elements that do not have a levee assigned can also be edited with the *Levee Express Editor*. Select the element and click the *Activate* button.



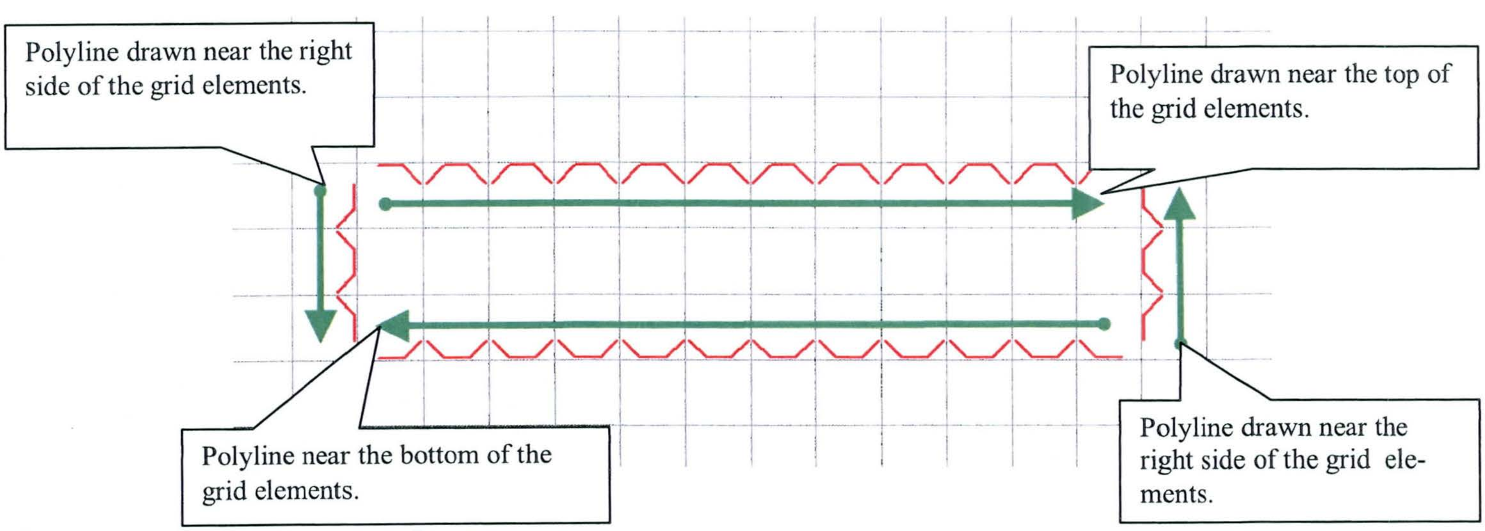
The following images represent the before and after pictures for creating the detention basin berm. Note that the levee does not go through the channel. Some grid elements do have both channel and levee components which is acceptable.

There is no natural inflow feature into this detention basin so it is acts more like a water tank. Opening a gap in the levee on the upstream side would create a proper detention basin. Deleting the levee in between the blue lines in the image on the right would open the detention basin to natural inflow. The blue lines are not part of the GDS. *Save* the *FLO-2D Files*. A lateral weir along the channel using a hydraulic structure could be used for detention basin inflow.

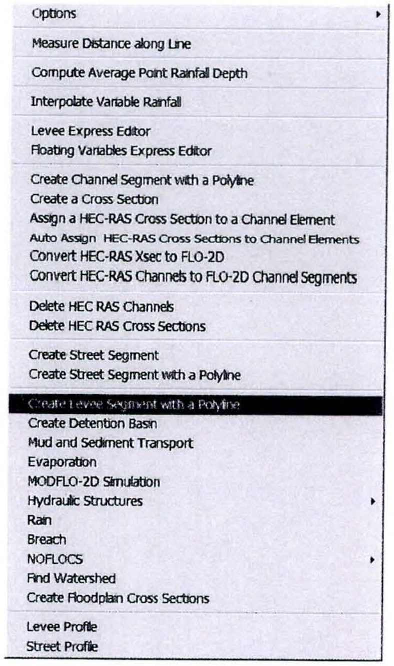


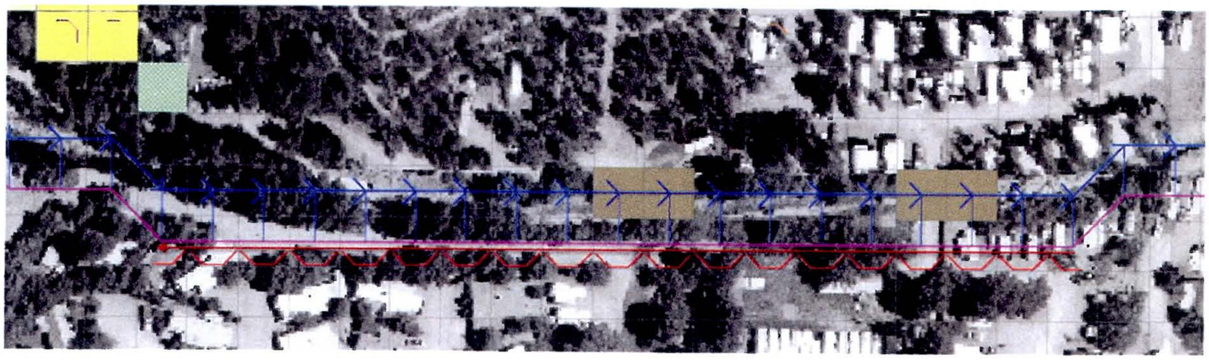
Step 4: Build a levee along the channel using a polyline.

When building a levee with a polyline, the cutoff direction of the levee is set by the proximity of the polyline to the side of the grid element. Test this process.

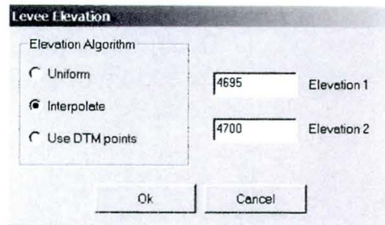


Zoom in on the central lateral section of channel north of the small development. Click *Tools/Create Levee with a Polyline*. Click along vertices to draw a polyline matching the *red arrow* in the following image. Double click the last vertex to finish the polyline. Keep the polyline near the bottom of the grid elements to block the south portion of the grid element.

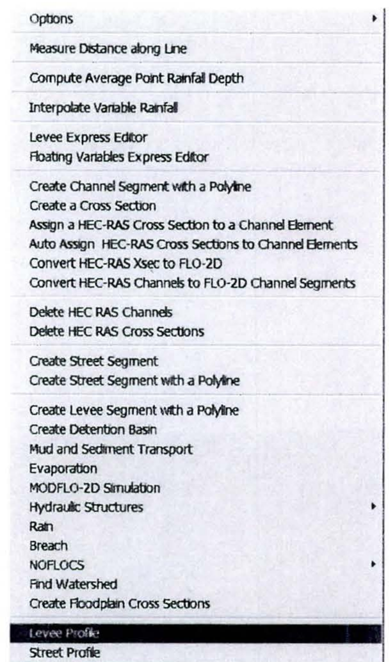


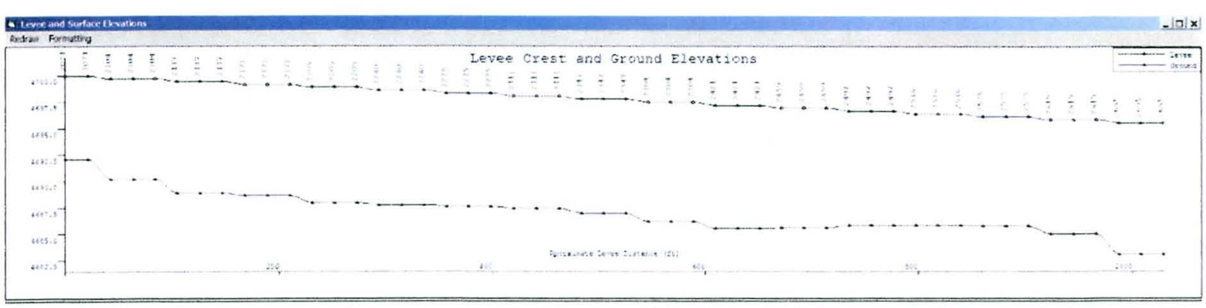


Fill out the *Levee Elevation* dialog box as shown below.



Click *Tools/Levee Profile* and note that the levee profile follows the slope of the topographic relief at a height of approximately 8 feet. Details can be added to the levee manually by using the *Levee Express Editor*. *Save* the *FLO-2D Files*.

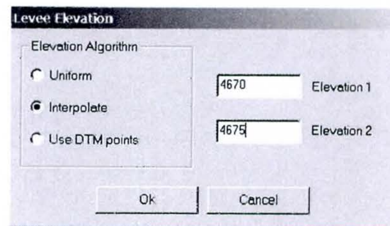




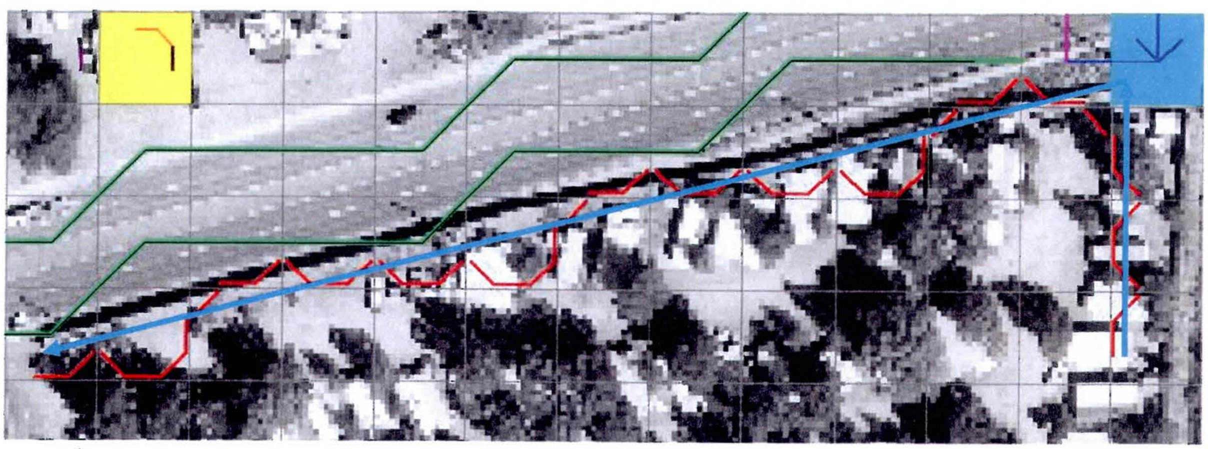
Step 5: Create a sound wall with a polyline.

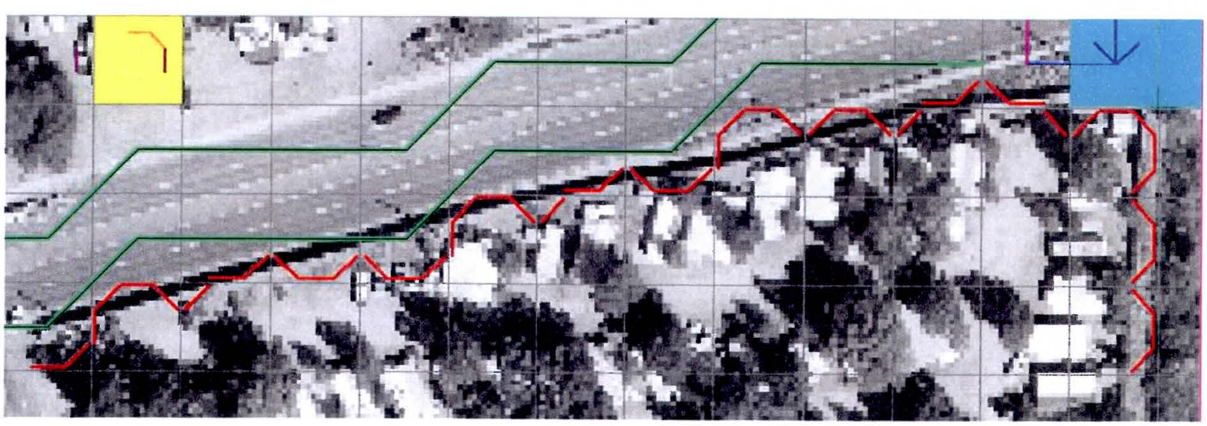
Zoom in on the housing development south of the highway. Use the *Tools/Create Levee with a Polyline* tool to draw the levee from South to North and East to West.

Interpolate the *levee crest elevation* from 4670 ft to 4675 ft.



Use the *Levee Express Editor* to adjust the levee placement to better match the wall. *Save* the *FLO-2D Files*. Use the Levee Express Editor to finalize the levee placement.

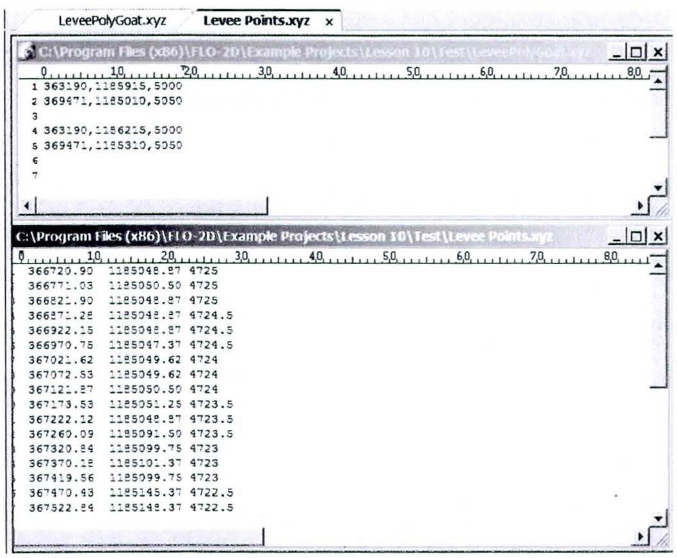




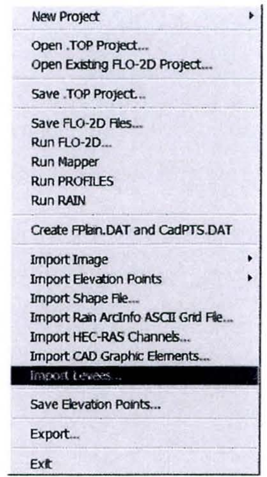
Step 6: Import a levee from a polyline.

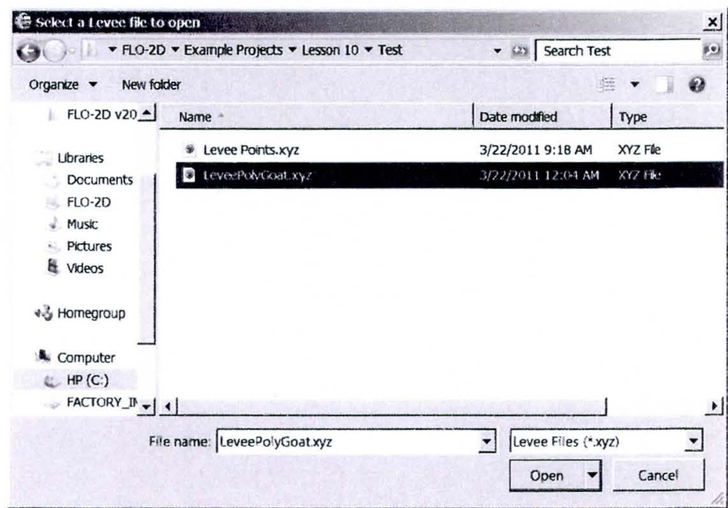
The GDS can create a levee from a 3-D polyline. The polyline is represented by a set of data points that represent the x- and y-coordinate and crest elevation of the levee. These files can be set up manually or with a GIS tool that captures the point elevation data from LiDAR data. The format is x y z and can be space or comma delimited. The following figure shows an example of two data files. The top figure contains two polylines with the end points and elevation data. The bottom file is a polyline with numerous vertices.

The GDS uses the point data and elevation to create a levee with an interpolated crest elevation for each grid element boundary. GDS also takes into account the position of the crest elevation points within the grid element so that the levee placement will align with the actual embankment feature.



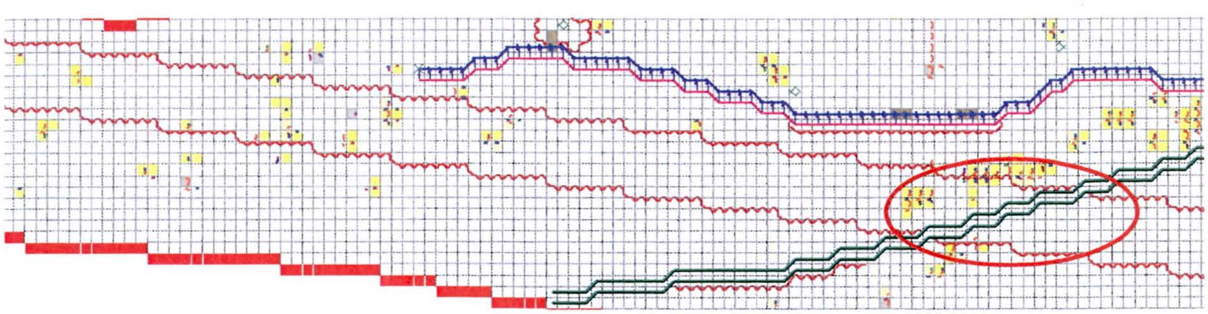
Click *File/Import Levee* and import the text file *LeveePolyGoat.xyz*. The GDS will automatically assign a levee to the grid element boundaries. Save the FLO-2D Data files.





Repeat the procedure for the *Levee Points.xyz* file.

This levee assignment method does not take into account other FLO-2D components. The levee will cross street elements, channel elements and buildings and therefore must be edited for those features. The following image show imported levees that were edited for the street.



Review Questions

1. List five features that can be modeled with the levee component.

____

____

____

2. True or False? A levee can share a grid element with a street or channel?
3. True or False? A levee can cross a street or channel?
4. The message "Levee crest elevation is lower than adjacent grid element elevation" appears. Describe how to fix this condition?

____

____

5. True or False? Flow overtopping levee is calculated using the weir equation?
6. Which output file reports if a levee was overtopped? ____________________
7. Name five different methods to input or edit levees.

____

____

____

____

____

8. Explain the output data in the LEVEEDEFICIT.OUT file. (Hint: see Data Input Manual)

____

____

____

9. Levee help documentation is found in which locations? Check all that apply.

- ☐ License Documentation
- ☐ PowerPoint Presentation
- ☐ Data Input Manual
- ☐ Pocket Guide
- ☐ Reference Manual
- ☐ Workshop Lesson Book

LESSON 11 – STREETS AND BUILDINGS

Overview

This lesson outlines the procedure for creating and editing streets and buildings in the GDS. It shows the user how to create steets with a polyline and how to define intersections. The automatic computation of arf/wrf factors using building polygons is also covered.

Required Data

The FLO-2D grid system files are needed for this lesson.

File	Content	Location*
*6 *.DAT files*	FLO-2D data files	*\Example Projects\Lesson 11*
Buildings.shp	Building shape file	*\Example Projects\Lesson 11*
Streets.shp	Streets shape file	*\Example Projects\Lesson 11*
Honolulu.jpg	Image file	*\Example Projects\Lesson 11*

**Typically these folders are in installed under C:\Users\Public\Documents\FLO-2D Basic Documentation*

Step-by-Step Procedure

To create streets and buildings using the GDS, follow these steps:

- Open the GDS program and load Lesson 11;
- Load aerial image;
- Load shape files;
- Edit layer properties;
- Review street data;
- Build and edit streets;
- Compute buildings from shape file;
- Review building data;

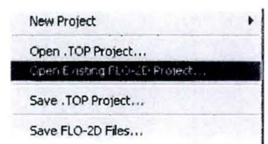
- Save and run.

Step 1: Open the Grid Developer System and load Lesson 11

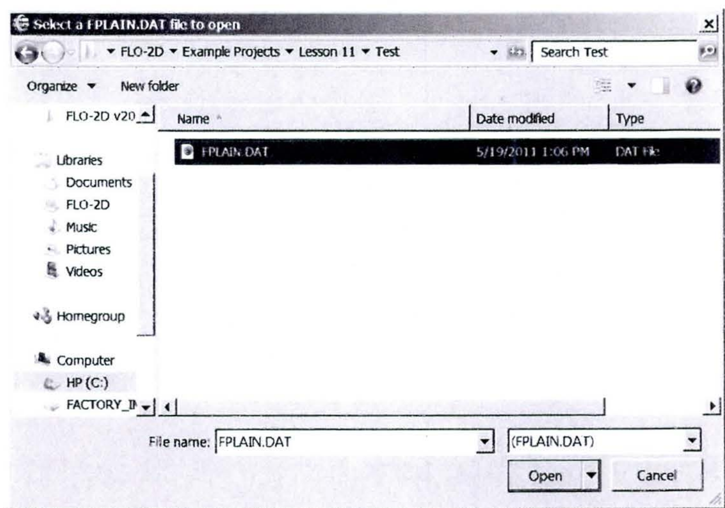
NOTE Using an Explorer window, make a new folder in the Lesson 11 folder named "TEST". Copy all the lesson 11 files to this folder and perform the lesson in the TEST folder. ***NOTE***

Double click the GDS Basic icon from the desktop. 

From the GDS *File* menu click *Open Existing FLO-2D Project…*

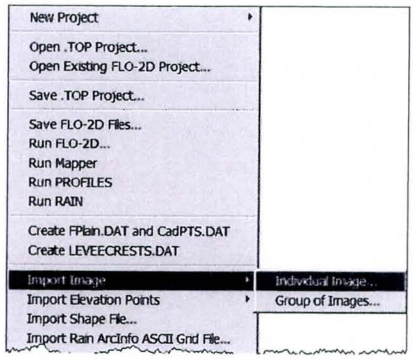


Locate the folder *Lesson 11/TEST* and select the *FPLAIN.DAT* file. Click *Open* to load the project.

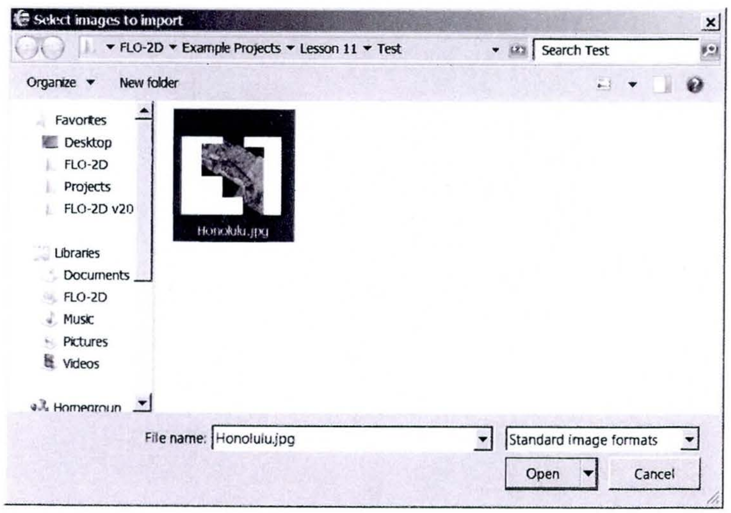


Step 2: Importing aerial images

To import a background aerial photo use *Import Image/Individual Image…* in the *File* menu.



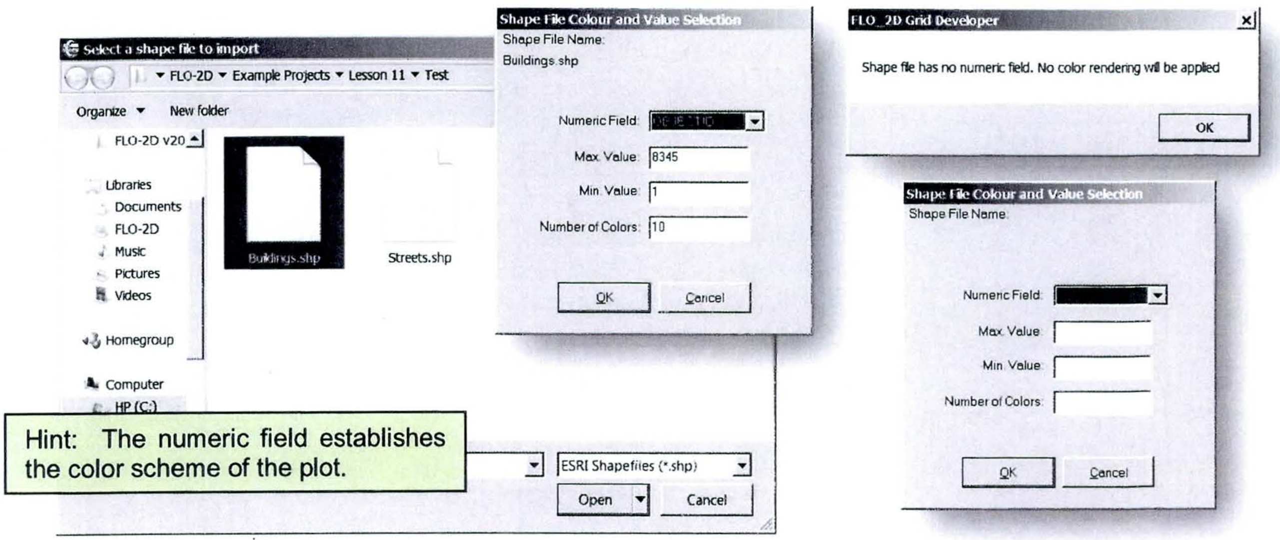
The image for this example project is located in the subdirectory *C:\ProgramFiles\FLO-2D\Example Projects\Lesson 11.* Images can be selected one by one or several at a time using the *Shift-click* or *Ctrl-click* on the files.



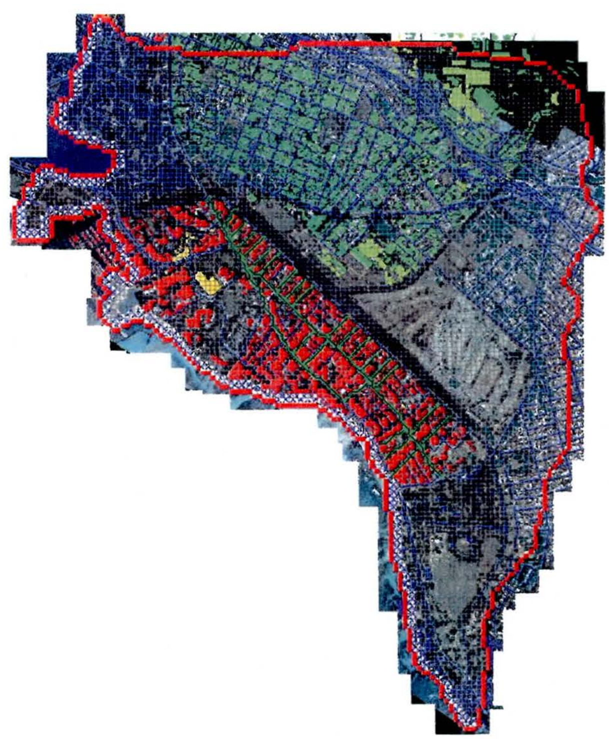
Step 3: Importing shape files

To import the building and street shape files use *Import Shape File...* in the *File* menu.

The shape files for this example project are located in the subdirectory *C:\ProgramFiles\FLO-2D\Example Projects\Lesson 11.* Shape files must be loaded one at a time. Click *OK* to load the shape files.

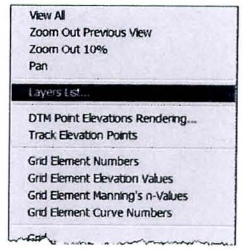


The grid system and and imported files are shown in the following image.

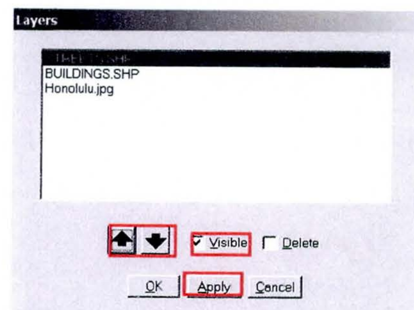


Step 4: Edit layer properties

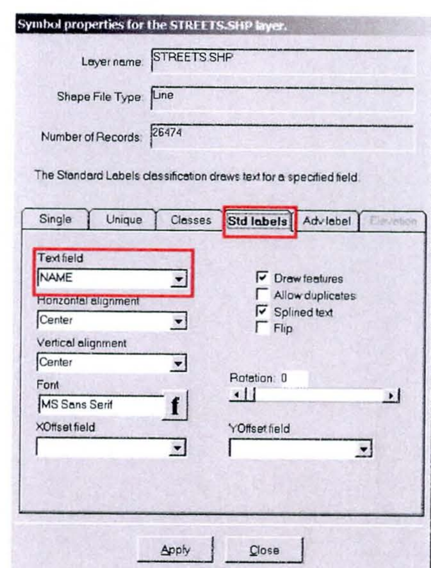
Click *View/Layers List...*



The *arrow* buttons are used to position the layers. Highlight a layer to move it up or down in the list. This determines the layer's heirachy.

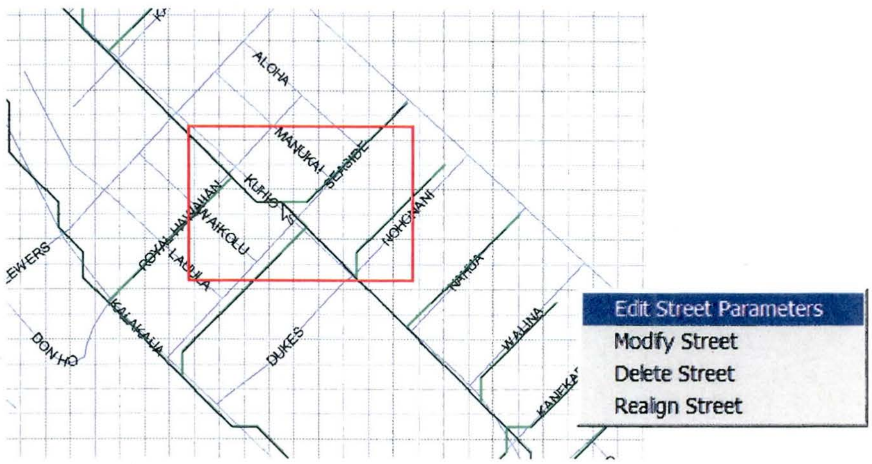


Make the *Honolulu.jpg* and *Buildings.shp layers* invisible by deselecting the *Visible* check box for each and click *Apply*. Double click the *Streets.shp* layer to access the *Symbol properties* dialog box. Select the *Std. Labels* tab and click *Apply*.



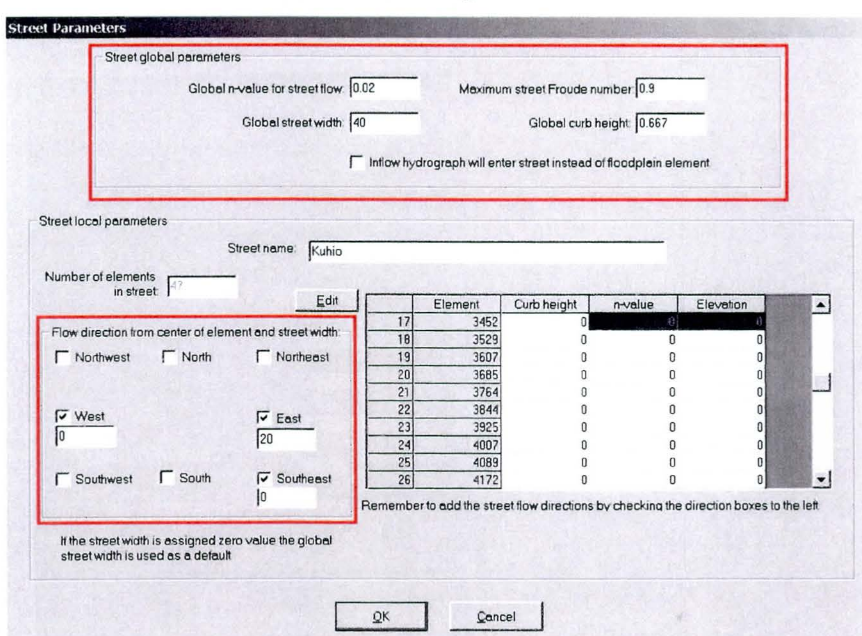
Step 5: Review street data

Position the mouse over any *street intersection* and click the *left mouse button*. The following image show the intersection of *Kuhio* and *Seaside*.



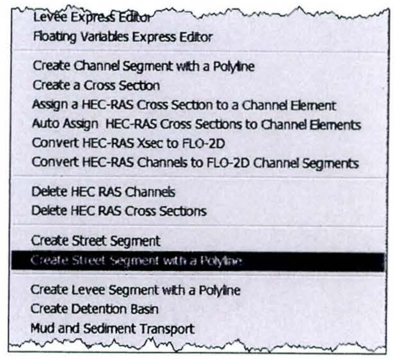
Click the *Edit Street Parmeters* option. The dialog box shows the global variables for the street segment. A *curb height, n-value and elevation* can be assigned for each street element and that value superceeds the global value.

Street intersections are defined using the *Flow direction from center and width* editor. When you click on an intersection, that location is loaded in the dialog box. Click *Cancel* to exit the window.

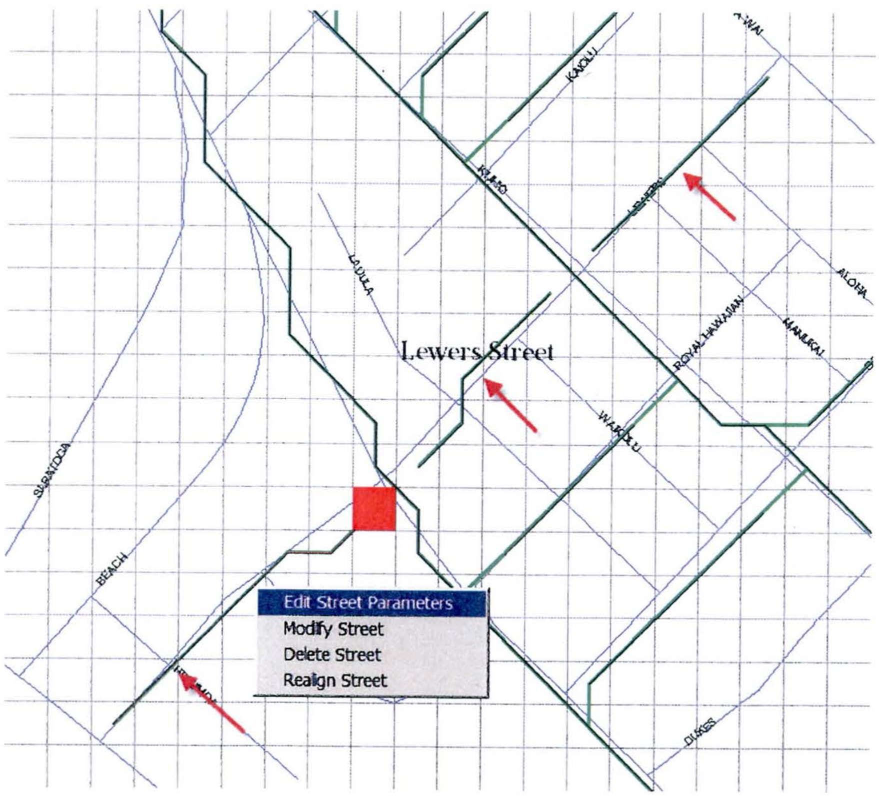


Step 6: Create street segments

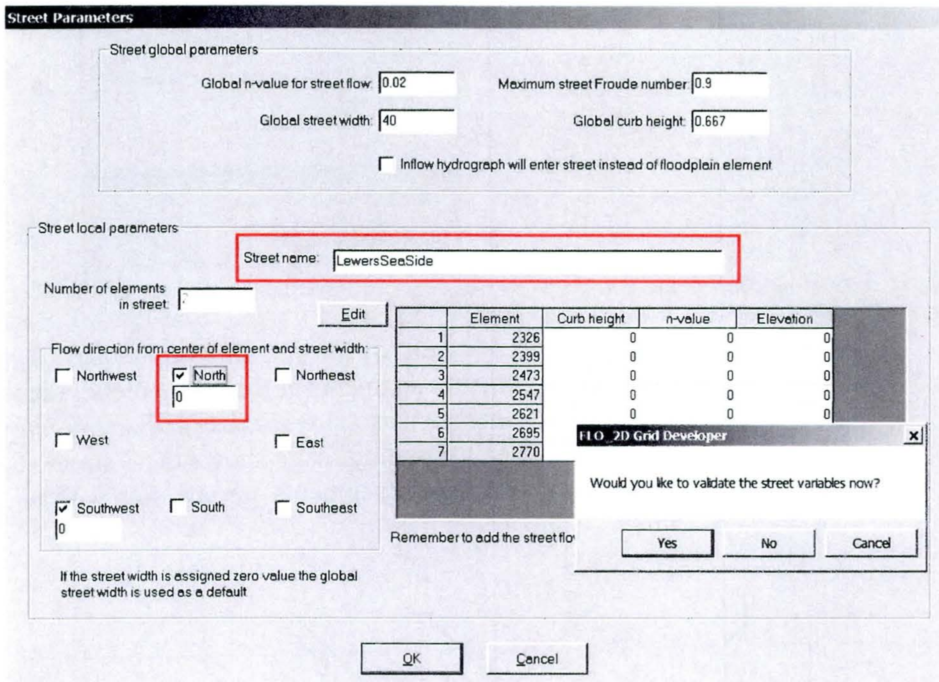
Click *Tools/Create Street Segment with a Polyline*.



Create a street segment for *Lewers*, *Liliuokalani* and *Uluniu* streets. Break the streets into segments at the side streets. Identify each street with a name such as *LewersSeaSide*, *LewersMiddle* and *LewersCitySide*. The following image shows the 3 street segments that represent *Lewers*. These segments are raw data with no intersections

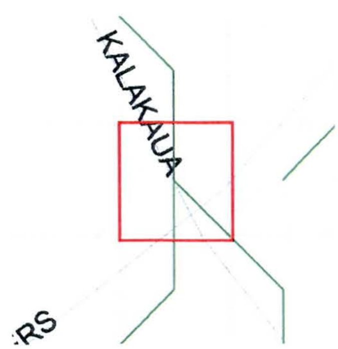


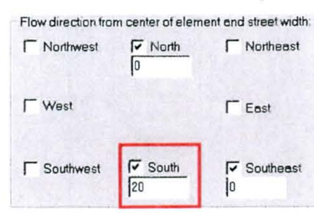
Edit the street data by clicking the *seaside Lewers segment* and clicking *Edit Street Parameters*. Enter a *Street name*. Highlight the *last grid element* and select the *North* flow direction to define the intersection.



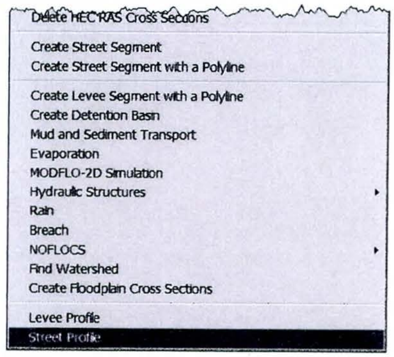
Click *OK* and *Yes* to validate the table.

Finish the intersection by clicking on the *Kalakaua street segment* and clicking *Edit Segment Parameters*. Select the *south flow directon* and enter a value of *20 ft*. Click *OK* and *Yes* to validate the table. Repeat this process for the next two segments.



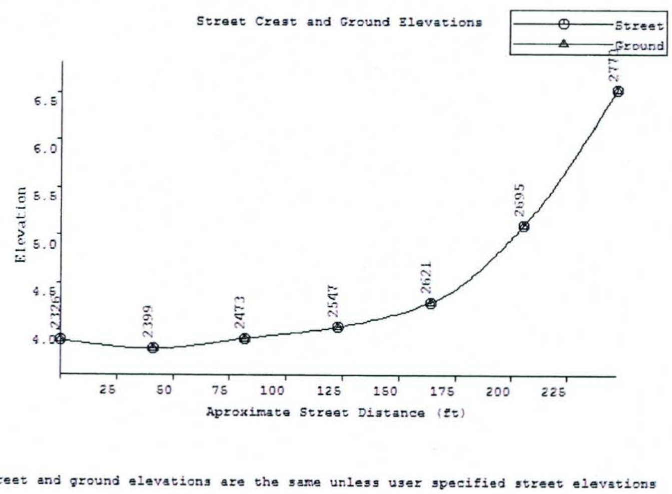


To see an elevation profile of the street click *Tools/Street Profile*.



Click the LewersSeaSide street segment.

Warning: Water will not flow freely down streets with high elevation spikes.

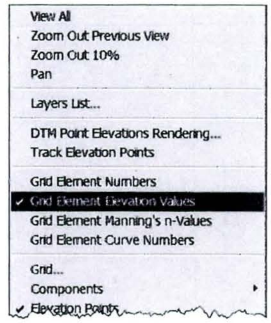


The street elevation is determined by the grid element elevation unless an elevation is assigned to each street element. Two methods are used to edit a street elevation:

1. Edit the floodplain element elevation
2. Enter a street element elevation

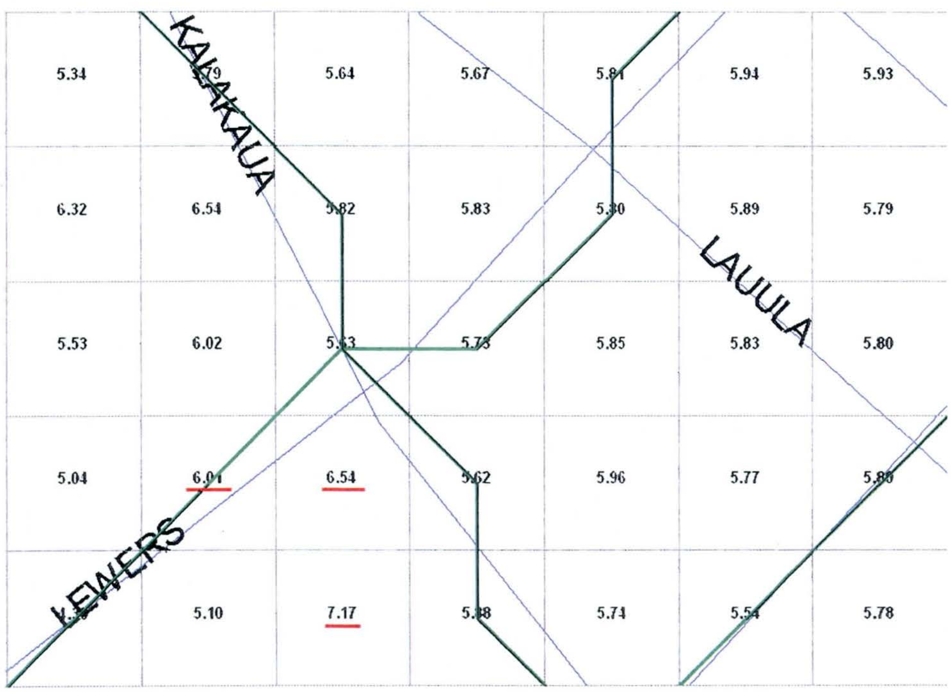
The elevation data used for Waikiki in this project was inadequate representation of the ground data. The street profile does not reflect the actual street elevations but an average value can be met by comparing the grid element elevations surrounding the streets and intersections.

Click *View/Grid Element Elevation Values*

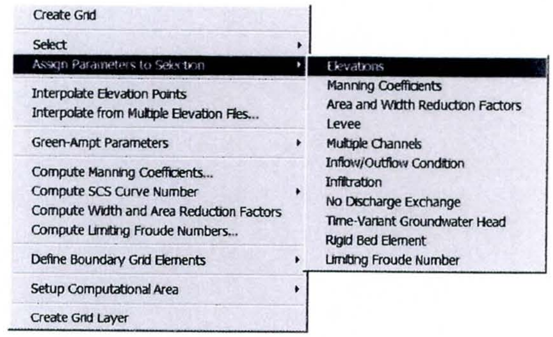


The grid elements underlined in red have an elevation that is higher than the mean. These elevations can be manually adjusted to more closely represent an actual elevation condition.

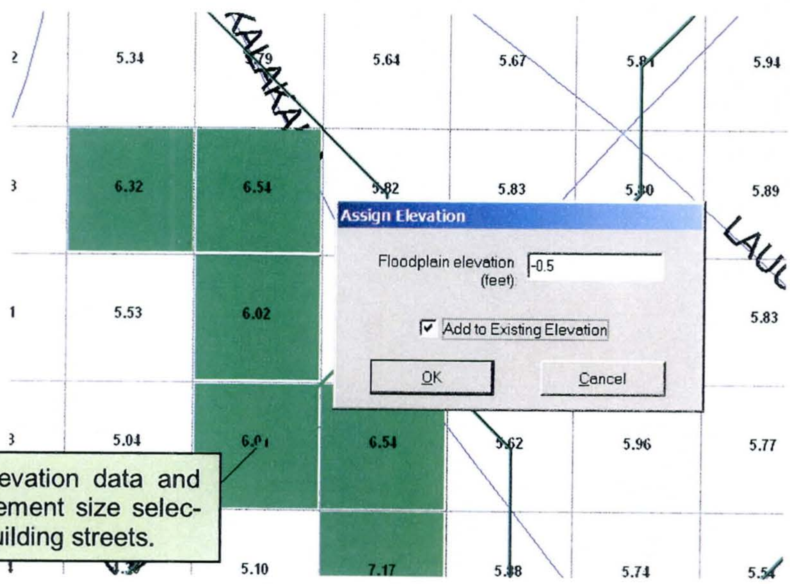
Editing the elevations corresponding to the streets will also make the street profile more realistic.



Click the *Select Element by Element* button. Select all the grid elements whose elevation can be lowered 0.5 ft. Click *Grid/Assign Parameters to Selection/Elevations*.



Enter the value *-0.50* in the *Assign Elevation* dialog box and select the *Add to Existing Elevation* check box.



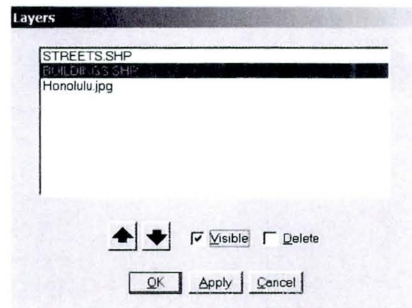
Note: Detailed elevation data and appropriate grid element size selection can facilitate building streets.

Perform this proceedure for the entire *Lewers* Street as well as the *Liliuokalani* and *Uluniu* streets to be built. Keep in mind that the overall objective is to make a realistic street profile by making small adjustments to the elevation data that better represent the actual elevation along the streets.

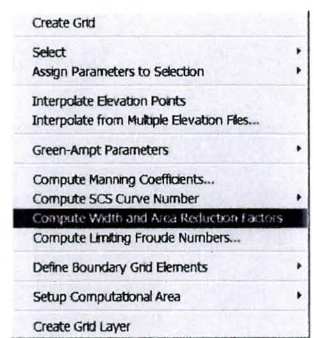
Save the project by clicking *File/Save FLO-2D Files...*

Step 7: Compute building data from shape file

Click *View/Layers List*. Select the *buildings* layer and check the *Visible* check box. Select the *Streets* shape file and uncheck the *Visible* check box. Click *Apply* and *OK*.

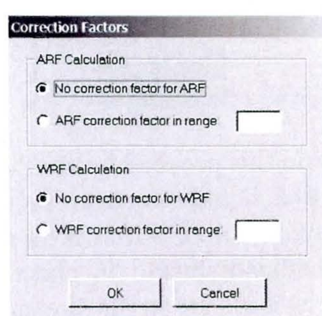


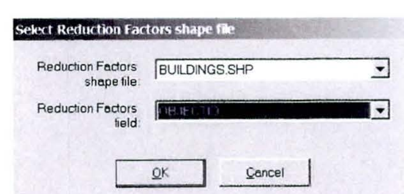
Click *Grid/Compute Width and Area Reduction Factors*.



Use the default setting with no correction factors and click *OK* and *OK* to select the *Buildings* shape file.

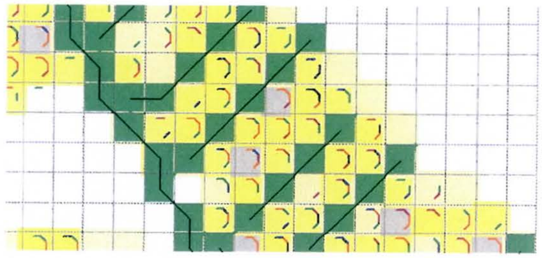
Note: If a correction factor were used, each ARF or WRF value computed would be mulitplied by the correction factor. For example, if a correction factor of 0.20 was selected the ARF or WRF value would be reduced by 20 percent.



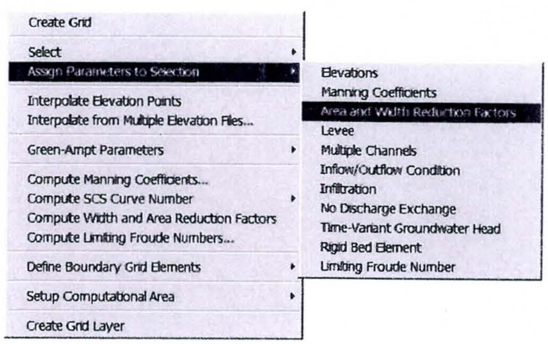


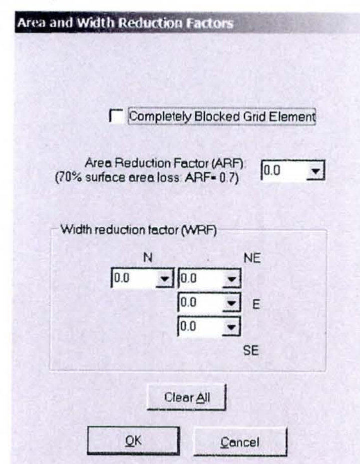
If it can be avoided, grid elements should not share components. It is advised to remove ARF and WRF components from street elements.

Select the elements along the streets that share an ARF/WRF component.

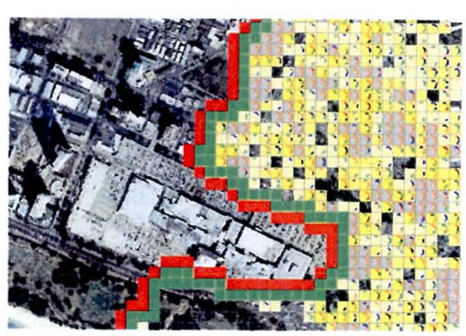


Click *Grid/Assign Parameters to Selection/Area and Width Reduction/Factors*. To remove the reduction factors from the selected grid elements, click the *OK* button with all values = *0.0*.



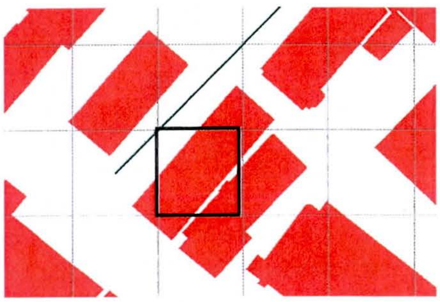


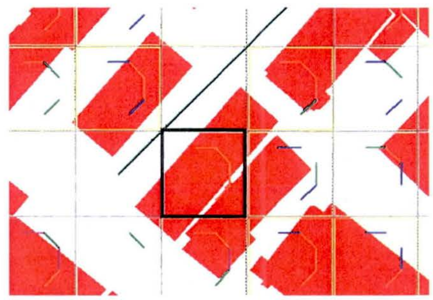
Apply the same technique for the outflow nodes.

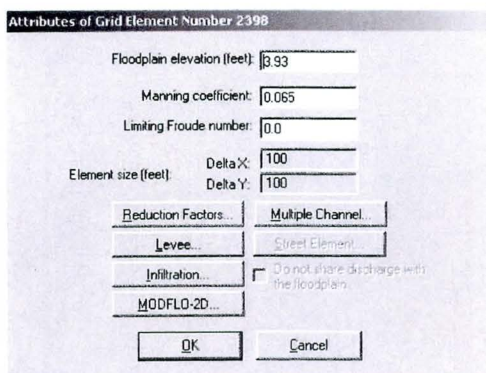


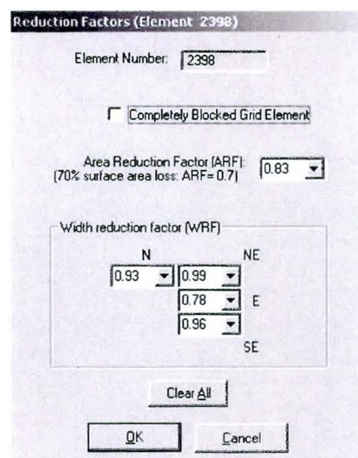
Step 8: Review and validate building data

The GDS computes the building data based on the colocation of the building polygon and the grid element. For example, if a grid element is covered by 50 percent of a polygon, the ARF value will be 50 Percent.









Review Questions

1. List five attribute of the street data.

   ____

   ____

   ____

2. True or False? A street segment name can contain spaces?
3. True or False? A street segment can cross a levee?
4. Describe how to delete a street segment?

   ____

   ____

5. True or False? Streets are represented as rectangular channels?
6. Can a street and a channel share a grid element? ____________________
7. Name three types of roadways that would NOT be modeled with a street component and explain why.

   ____

   ____

   ____

   ____

   ____

8. Explain how to view and edit street profile elevation. (Hint: see Data Input Manual)

   ____

   ____

   ____

9. Can two street segments share a grid element? ____________________