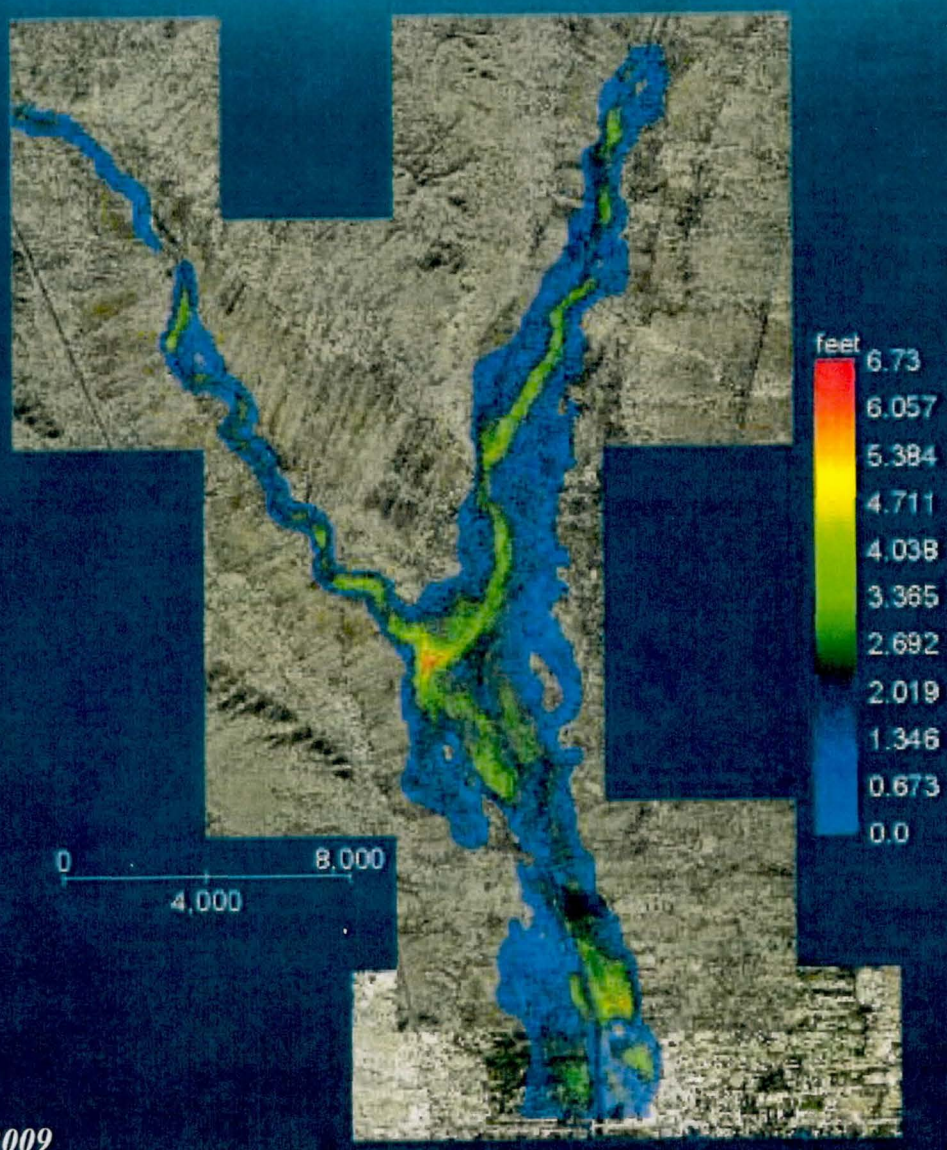
FLO-2D

Reference Manual



Version 2009

FLO-2D

Pocket GUIDE

Version 2009.06

Contact Information

General

FLO-2D
P.O. Box 66
102 County Road 2315
Nutrioso, AZ 85932
Phone: (928) 339-1935
www.flo-2d.com
contact@flo-2d.com

Purchasing

Karen O'Brien
Karen@flo-2d.com
(928) 339-1935

Technical Support

FLO-2D - Jimmy O'Brien
Jim@flo-2d.com
(928) 339-1935

GDS and MAPPER - Reinaldo Garcia

Reinaldo@flo-2d.com

Table of Contents

Contact Information ii

Resources 1
- Manuals 1
- Lessons and Tutorials 2
- Power Point Presentations 3
- Website - www.flo-2d.com 3
- FLO-2D License 3
- ESRI MapObjects® GIS 4
- ESRI Runtime Engine® for Mapper.NET 4

FLO-2D Suite of Programs 5

Set-up 6

General Overview 7
- The Basics 7
- New Features and Enhancements 7
- Porting From Version 2007 to Version 2009 8

Getting Started 11
- What data is required? 11
- Before you start 12
- Estimate the project area 13
- Selecting the grid element size 13

Start simple, then add detail 15
Saving data 16

Building a Project 17
Some General Guidelines 18

Channel Hints and Guidelines 21
Overview 21
Creating the Channel Data File 21
Additional Channel Data Instructions 24

Modeling Guidelines 28
Some Basic Data File Checks 28
Data Errors 29
Troubleshooting: Is the flood simulation running OK? 30
Make some adjustments 35
A few important things to consider 38
How can I improve model speed and stability? 40

Making Flood Maps 45

Resources

Manuals

New FLO-2D Models come with a hard copy of the manuals as well as an electronic copy on the CD. Updates and Annual Subscriptions include only the electronic version of the manual. The available manuals include:

FLO-2D Reference Manual

The Reference Manual provides an overview of 2-dimensioal flood routing. Modeling theory and the FLO-2D system are discussed. There are some instructional comments and modeling parameter data including sediment transport, roughness and infiltration.

Data Input Manual

The data input manual contains descriptions of all modeling input variable parameters. There are instructions for installation and getting started as well as for using the pre-processor and post-processor programs. The manual also contians information about data limitations, output files and a trouble shooting.

GDS Manual

This manual contains a description of the GDS components and all the graphic tool commands. It also has a getting started section to help set up a grid system for an overland flow model and a detailed description of creat-

ing a rainfall simulation.

Mapper Manual

This manual contains a description of the MAPPER components and tools. It has detailed instruction on creating hazard maps.

Mapper_Net Manual

This manual contains a description of the Mapper_Net components and tools.

Lessons and Tutorials

The FLO-2D Software package comes with workshop lessons and tutorials to help the user utilize the many components and features. These include:

Workshop Lessons

Lesson 1: Getting Started GDS
Lesson 2: Detailed Grid System Attributes GDS
Lesson 3: Channel GDS
Lesson 4: HEC-RAS Cross Section Conversion
Lesson 5: PROFILES
Lesson 6: MAPPER

Tutorials

1. FLOENVIR Rainfall Tutorial
2. FLOENVIR Street Tutorial
3. GDS Create a Detention Basin Tutorial

Power Point Presentations

All Power Point presentations that are used in FLO-2D short courses are available on the model CD. These Power Point presentations cover almost every feature of the FLO-2D modeling system

Website - www.flo-2d.com

The FLO-2D website contains extensive information about flood modeling. You can also purchase models, updates and subscription services.

Updates to the FLO-2D programs are posted at the website frequently (at least once a month). We recommend that you check for new updates to the FLO-2D model and its processor programs prior to starting a new project.

FLO-2D License

The FLO-2D model license can be reviewed in the FLO-2D subdirectory. Basically it states that the FLO-2D model can be loaded on any computer in the office of purchase. The license permits the use of the software by the licensee or its employees on any and all computers located at the office of purchase. The license does not include the right to copy or distribute the FLO-2D Licensed Software outside the office of purchase. The license does not permit the use of the FLO-2D model on a laptop or portable computer outside of the office of purchase.

ESRI MapObjects® GIS

The GDS and Mapper programs have ESRI MapObjects® GIS controls embedded in the programs. One ESRI MapObjects® user license is granted with each purchase of the FLO-2D model. The GDS or Mapper programs with the MapObjects® tools can only used on one computer at a time. For most offices this use limitation should not restrict application of the GDS or Mapper processor programs as these programs are only required when creating and editing the grid system attributes or viewing the FLO-2D model results. If you anticipate that you will have multiple computers with the GDS and Mapper in use at the same time, you can purchase additional MapObjects® licenses at the FLO-2D website.

ESRI Runtime Engine® for Mapper.NET

Mapper_NET_2009 will create high resolution flood hazard contour maps. It is integrated with ArcGIS Engine and has unique mapping features and tools including break-lines to control contour generation. The Mapper_NET installation requires having a license for both ArcGIS® desktop and ArcGIS® 3D Analyst. ArcGIS Engine Runtime 9.2 must also be obtained from ESRI and installed. To install the ArcGIS Engine Runtime 9.2, the .NET Framework 2.0 must already available on the computer. If it is necessary to install ArcGIS Engine Runtime 9.2 on your computer, see the document Mapper_NET Installation.doc.

FLO-2D Suite of Programs

FLO-2D

Two dimensional floodrouting model for river and unconfined flooding.

GDS

Graphical interface program for creating FLO-2D grid systems and for graphically editing model data.

MAPPER

Mapping software to display FLO-2D results.

*River*FLO-2D

A high resolution, finite element flood routing model for in-channel 2-D flows.

MAPPER_NET

Integrated GIS mapping program with higher resolution map controls.

DFIRM

This software generates FEMA submittal-ready DFIRM maps directly from the FLO-2D results.

FreqPlot

Flood peak discharge and rainfall frequency analysis software that allows comparison of different frequency distributions.

Set-up

Computational Requirements

FLO-2D is compatible with the MS-Windows™ operating systems including the newer versions of Windows Vista®. Recommended minimum computer requirements are 4 gb RAM and 500 MB of available hard disk space. Projects with a large grid system (> 100,000 cells) can put a significant demand on computer resources.

Installation

To install the FLO-2D software package onto the computer hard drive, load the CD or copy the installation files to your computer and wait for the installation to begin. If the installation wizard does not automatically start, locate the SETUP.EXE file with an explorer program and double click on it. The default installation subdirectory for the model files is *C:\Program Files\FLO-2D*. The FLO-2D model, the manuals, tutorials, workshops, powerpoint presentations and all the processor programs are loaded into the FLO-2D subdirectory. The final step of installation is software activation. If you did not receive an activation key, email us at activation@flo-2d.com.

Un-install

You can remove the FLO-2D program and all of its attendant software from your computer using the standard

Windows Remove Program procedure. From the control panel, run the Remove Programs utility and remove FLO-2D. The FLO-2D folder including example projects and flo_help will still be in the Program Files folder. To complete the un-install process, delete the FLO-2D subdirectory from the Program Files Folder.

General Overview

The Basics

FLO-2D was written in Fortran 95 computer language. There are other coding languages embedded in the model. Depending on your computer speed, project application and flood duration, the flood simulation might have a runtime ranging from 5 minutes to more than a day. The code has been optimized for both 32-bit and 64-bit multiple processor computers. Virtually any Windows computer will suffice, but faster and bigger is better.

There are several processor programs that can be used to graphically create or edit spatial variability in the data base. To generate the basic data files and graphically edit the data, the grid developer system (GDS) program should be used. The FLOENVIR and PROFILES programs are also use for data editing. The graphical user interface (GUI) was developed to simplify the ASCII text data input and access the FLO-2D system.

New Features and Enhancements

New model components and features are added to the FLO-2D model system with each new release approximately every 15 to 18 months. Between new releases, updates, minor enhancements and bug fixes are posted to the website, typically on a monthly basis. New features may include new components such as MODFLOW groundwater modeling, dam or levee breach erosion, or optimization for multiple processors. See the website for the new features in the next version.

Porting From Version 2007 to Version 2009

There are very few changes to convert Version 2007 data files to the Version 2009. Except for initialization of the groundwater model MODFLOW switch in CONT.DAT and the new CHAN-BANK.DAT file, all the changes involve optional data or components. The optional data file revisions do not require any 2007 data files modifications for the FLO-2D model to run.

CONT.DAT

Surface water/groundwater interaction can now be simulated at runtime through the integration with the MODFLOW groundwater model. The new switch parameter to initiate the groundwater model is IMODFLOW at the end of line 3 after the ISED parameter. For the "no groundwater" default set IMODFLOW = 0.

For surface water/groundwater modeling, review the required MODFLOW parameters in the MODFLOW. DAT file in the Data Input Manual.

CHAN.DAT

The most important model revision is the elimination of the IQDIR channel direction from the CHAN.DAT file. The data input change was automated in the GDS and FLOENVIR. The required data revision is as follows:

Delete IQDIR from Line 2. This can be done manually, but it is only necessary to open and save the CHAN. DAT in the GDS or FLOENVIR and the IQDIR vari-

able will be eliminated. A new data file CHANBANK.DAT is generated to replace the IQDIR parameter. This file contains the left and right bank channel elements and it is automatically generated when the channel is saved in the GDS and FLOENVIR. After the CHANBANK.DAT is generated, it may be necessary to make some minor adjustments to the automated selection of the right banks. Again this can be done graphically in the GDS or FLOENVIR.

Optional Data File Revisions

Levee fragility curves can be assigned in the BREACH.DAT file and the global and individual fragility curve assignment to the levee elements is accomplished in the LEVEE.DAT file. The new levee fragility curve data is appended to the each file. Refer to the respective data file descriptions to input the fragility curve parameters.

Additional Output Files

Three new output files are now generated:

FPINFILTRATION.OUT - Total cumulative infiltration by grid element.

VELTIMEC.OUT and VELTIMEFP.OUT - Maximum velocity and time of occurrence in descending order for the channel and floodplain respectively.

Porting data files from earlier versions of the FLO-2D model

Data files from model version 2004 and earlier requires that the data files first be updated to Version 2006 format to be able to make the simple changes to Version 2009. For data files versions prior to 2006, it is suggested that the data files be edited directly by referring to the Version 2009 Data Input Manual and using an ASCII text editor. There is a program available from the website www.flo-2d.com to convert data files from Version 2004 to Version 2006. As a final check, it recommended that the data files be compared with the Version 2009 Data Input Manual.

Getting Started

What data is required?

DTM Data

To start a FLO-2D model outline the project area and compile available mapping, imagery and digital terrain model (DTM) data. The imagery and DTM points must have the same coordinate system. If aerial imagery or digital topo maps are not provided with the project, you may be able to purchased then through the internet. The most common formats for digital imagery are *.tif, *.sid and *.jpg files and these must have corresponding world files (e.g. *.tfw, *.sdw and *.jgw). If photogrametric or LiDAR data are not available, DEM data can be used. Elevation data formats that are accepted by the GDS are ASCII x y z data sets and elevation shape files.

Hydrologic data

Hydrologic data for flood simulation can include both rainfall and discharge hydrographs. These data bases can usually be obtained from the local, state or federal agencies. It is suggested that the hydrologic data be carefully reviewed because the flood volume will determine the area of inundation. The user must also decide whether infiltration losses will be simulated.

Floodplain and channel detail

If river cross sections, bridges, culverts, buildings and streets are to be simulated, the user must be able to locate these features with respect to individual grid elements. Aerial imagery is invaluable for this purpose. Component data may be required for these components. Bridges and culverts will need rating curves or tables. Streets will need width and curb height data. River cross sections may have to be surveyed.

Before you start

- Review the following pages in this section completely.
- Review the FLO-2D License Agreement on page iii of the FLO-2D Users Manual to understand your options for installing the program on multiple computers.
- Work through the tutorials and workshop lessons that pertain to your project.

Updates

When starting a new FLO-2D project, the first step is to visit the website www.flo-2d.com and download any model, processor program, manual or document updates. New features are added to the programs throughout the year. Programming bugs are fixed as they are identified. Do not hesitate to notify us of any bugs that you encounter and we address as soon as possible. Program revisions and bug fixes are listed on the web site in the FLO-2D Model Revisions.document by date.

Tutorials

The tutorials and lessons located in the FLO-2D help folder (FLO-2D\flo_help) will assist you to develop detailed flood components. Please check the website to download any newly posted tutorials/lessons.

Estimate the project area

One of the keys to setting up an efficient model is to determine the project area and estimate a desired grid element size. The project area should be located so that it is not affected by either inflow or outflow conditions. The inflow and outflow nodes should be considered as non-essential nodes (sources and sinks) and these should be located outside the project area.

Selecting the grid element size

Once the overall project area has been identified, estimate the grid system size (as a rough rectangle) and determine the approximate number of grid elements that would be required for different size grid elements such as 50 ft, 100 ft, 200 ft, etc. The grid elements are square and estimating the maximum number of grid elements is relatively easy.

Selecting the grid element size will control how fast your FLO-2D flood simulation will run. FLO-2D users often choose a grid element that is smaller than necessary. A small grid element combined with a high flood discharge can result in long flood simula-

tions times.

To help with the grid element size selection, the following criteria are suggested: The estimated peak discharge Q_{peak} divided by the surface area of the grid element A_{surf} should be in the range:

$$0.1 \text{ cfs/ft}^2 < Q_{peak}/A_{surf} < 1.0 \text{ cfs/ft}^2$$

or in metric:

$$0.03 \text{ cms/m}^2 < Q_{peak}/A_{surf} < 0.3 \text{ cms/m}^2$$

The closer Q_{peak}/A_{surf} is to 0.1 cfs/ft^2 (0.03 cms/m^2), the faster the model will run. If the Q_{peak}/A_{surf} is greater than 1.0 cfs/ft^2 or 0.3 cms/m^2, the model should be expected to run more slowly.

After the grid element size has been selected, proceed with establishing the grid system using the GDS. There are GDS workshop lessons to assist you in getting started on a new project.

GRID SYSTEM SIZE	
Number of Grid Elements	Model Simulation Speed
1,000 – 15,000	Very Fast (minutes)
15,000 – 30,000	Fast (~hour)
30,000 – 60,000	Moderate (hours)
60,000 - 150,000	Slow (up to one day)
> 150,000	Very Slow (a day or more)

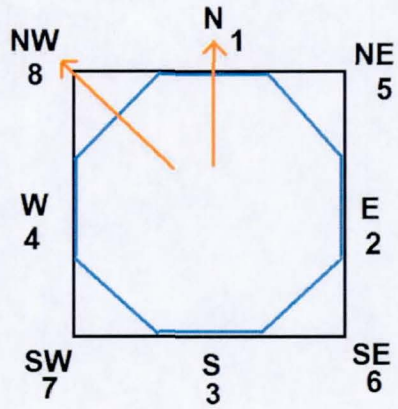
Start simple, then add detail

The first flood simulation for any project will be a simple overland flow model upon which a more detailed flood simulation will be gradually built. A suggested order of component construction is as follows:

- Rainfall/Infiltration
- Channels
- Levees
- Streets
- Buildings
- Hydraulic Structures (culverts, weirs and bridges)
- Multiple Channel (rills and gullies)
- Mud and debris flows/sediment transport
- Evaporation
- Floodways

As new components are added to a model and tested, other components switches can be turned off in the CONT.DAT file.

FLO-2D routes flows in eight directions as shown in the following figure. The four compass directions are numbered 1 to 4 and the four diagonal directions are numbered 5 to 8. Some components such as levees are placed on boundaries of the grid element. The grid element boundaries create an octagon in this case.



Saving data

When you are creating or editing the data files, it is suggested that you save the data files frequently and use one folder for testing your project and one for editing your project.

Building a Project

Create a Project Folder

Start by creating a subdirectory for the project data files and import the DTM data base files, map images and aerial photos.

Build the Project Files

Use the GDS to build a grid system. Most data files can be graphically created in the GDS. You can follow the GDS "Getting Started" lesson to initiate a project. For easy access, put the GDS icon on the desktop.

Run the FLO-2D model

Once the six required basic data files have been created (CADPTS.DAT, FPLAIN.DAT, CONT.DAT, TOLER.DAT, INFLOW.DAT and OUTFLOW.DAT), an overland flood can be simulated. You can run a FLO-2D simulation by:

1. GUI - click on the "*Execute*" pull down menu.

2. GDS - click on "*Run FLO-2D*" command in the File menu.

3. Double click on "*FLO.EXE*" in the project subdirectory. Put "*FLO.EXE*" in the project folder first.

Some General Guidelines

Data Input

When the data format seems confusing, review the example project data files provided in the Example Projects subdirectory of the FLO-2D folder.

File Management

The output files in the project folder will be overwritten during subsequent model runs. To save any output files that might be overwritten, rename the file or create a new folder, copy all the *.DAT files into it and then run the new flood simulation in that folder.

Graphics Mode

To view a graphical flood progression over your project flow domain, follow these steps:

1. Click the GUI (FLO-2D icon) to turn 'on' the graphics switch (LGPLOT = 2) in the CONT.DAT file form. Set Graphics Display to 'Detail Graphics.'
2. Assign an update screen refresh time (GRAPHTIM) in the lower left hand corner of the CONT.DAT file form to 0.05 or 0.10.

Simulating Channel Flow

To add a main channel to an overland flood routing routine, follow this procedure:

1. Review workshop lesson 4 and 5.
2. If surveyed cross section data is available, create

the XSEC.DAT file first. Then generate the CHAN.DAT file in the GDS.

3. Interpolate the cross section data in the GDS or in PROFILES.
4. Set the 'Main Channel' check box switch (ICHANNEL = 1) in the CONT.DAT file.
5. Prepare any channel inflow hydrographs for the INFLOW.DAT file.
6. Select a channel inflow hydrograph to be plotted (IDEPLT) in INFLOW.DAT file.
7. Assign channel outflow node(s) in OUTFLOW.DAT.
8. Review the "Channel Hints and Guidelines" section.

Mud and Debris Flow Simulation

Simulating mud and debris flows requires additional data:

1. 'Turn On' the MUD switch option check box in CONT.DAT file.
2. Turn off the XCONC and ISED (sediment transport) switches in the CONT.DAT file form.
3. Create the M-line (Line 1) in the SED.DAT file.
4. Assign sediment concentrations or volumes to the inflow hydrographs in the INFLOW.DAT file.
5. Review the document, "Simulating Mudflow Guidelines" in the FLO-2D/flo_help folder.

Mobile bed simulation is complicated and should be attempted only after a rigid bed model is fully functional:

1. Turn 'on' sediment transport switch (ISED = 1) in the CONT.DAT file form.
2. Turn off the mudflow switch (MUD = 0) and set XCONC = 0.00 in the CONT.DAT file form.
3. Assign the SED.DAT file sediment parameters.
4. For channel sediment transport, set ISEDN = 1 for each channel segment in CHAN.DAT.
5. Read the 'Sediment Transport – Total Load' section in the FLO-2D Users Manual.

The following units are used in the FLO-2D model. These are the units that are entered in *.DAT input files.

English Metric Conversion		
Variable	English	Metric
discharge	cfs	m^3/s (cms)
length (depth)	ft	m
hydraulic conductivity (infil)	inches/hr	mm\hr
rainfall and abstraction	inches	mm
soil suction	inches	mm
velocity	fps	mps
volume	acre-ft	m^3 (cm)
viscosity	poise (dynes-s/cm^2)	poise
yield stress	dynes/cm^2	dynes/cm^2

Channel Hints and Guidelines

Overview

Review the CHAN.DAT file description in the Data Input Manual. It includes a list and explanation of all channel variables and instructions on how to apply them.

The most important aspect of simulating channel flow is to correctly balance the slope, flow area and roughness to replicate field conditions. In the FLO-2D model, river flooding using the channel component is simulated as one-dimensional, depth averaged flow. Each channel element is represented by either rectangular, trapezoidal or surveyed cross section. Simulating river flow requires the following data:

- Location of the channel with respect to the grid system;
- Channel roughness;
- Length of channel within the grid element;
- Channel cross section data.

Channel slope is computed as the mean bed elevation difference between the channel elements. Channel elements must be contiguous to be able to share channel discharge.

Creating the Channel Data File

The procedure for creating a river channel simulation is as follows (see Channel and PROFILE Lessons):

Select Channel Cross Sections

River cross section survey data is organized in the XSEC.DAT file with a station and bed elevation format. Each cross section may represents one or more channel elements. Each channel element is assigned a cross section in the CHAN.DAT. For channel design projects, a rectangular or trapezoidal cross section may be selected.

Locate the Channel Element with Respect to the Grid System

Use the GDS processor programs to identify the left bank channel element (see the GDS Channel Lesson 4). For flow to occur through a river reach, the channel elements must be neighbors.

Assign the Preliminary Channel Data

The GDS will list the selected channel elements in a dialog box. Preliminary channel data such as shape, channel element number, channel extension direction, roughness n-value, channel length and cross section number can be assigned in the channel editor dialog box.

Define the Right Bank Element

The channel width can be larger than the grid element. For example, a channel may be 1000 ft wide while the grid element is only 200 ft wide. The left and right bank elements can be separated by several grid elements. The channel component interacts with the right and left bank elements to share discharge with the floodplain. Each bank element can have a unique top-of-bank elevation.

Assign the Cross Section Number

Assign the surveyed cross section number in XSEC. DAT to the corresponding channel element in CHAN. DAT. Assign a zero (0) value to the rest of the channel element cross section numbers. Typically, there are only a limited number of cross sections and many channel elements. Before interpolation, therefore, the channel profile will look like a staircase. After you click on the GDS 'Interpolation' button each channel element will have a unique cross section and bed elevation and the cross sections numbers in XSEC.DAT and CHAN.DAT will correspond and will be renumbered from top to bottom starting with cross section number 1. Both the slope and cross section shape for each channel element will be interpolated between those channel elements with assigned cross sections.

Assign the Channel length Within the Grid Element

The channel length (XLEN) within a grid element is estimated. The GDS calculates the XLEN automatically. The channel lengths are then summed and reported by GDS for each segment. The river center-line distance can be estimated with the GDS Measure Distance along a Line tool. The individual XLEN values can then be adjusted so that the reach length is exact.

Adjust the Channel Bed Slope and Interpolate the Cross Sections

The cross section geometry and slope can be re-inter-

polated between any two channel elements in the PROFILES program. The result of this interpolation is an adjusted cross section shape and bed slope. The assigned surveyed cross sections retain their original shape and elevations. The bed slope for rectangular and trapezoidal can also be adjusted.

Assign the Manning's n-value.

Initially a uniform Manning's n-value can be assigned to all the channel elements. Using the limiting Froude number (FROUDC) in Line 1 of the CHAN.DAT file, spatially variable n-values can be adjusted. The n-value should represent a composite flow resistance for the entire channel including bed irregularities, obstructions, vegetation, variation in channel geometry, channel expansion and contraction, potential rapidly varying flow and variable river planform. Poor selection of n-values (particularly underestimating n-values) or failure to provide spatial variation in roughness can result in numerical surging.

Additional Channel Data Instructions

The user can select several options when setting up the channel data file including grouping the channel elements into segments, specifying initial flow depths, identifying contiguous channel elements that do not share discharge (NOFLOC's), identifying channel elements that don't share discharge with the floodplain (NOEXCHANGE), assigning limiting Froude numbers and specifying

depth variable n-value adjustments. These options are discussed in more detail in the CHAN.DAT file description in the Data Input Manual. A few instructional comments follow:

- Dividing the channel into segments by cross section geometry may facilitate organizing and reviewing the results. For example, a segment may represent a tributary or a concrete section of the main channel. Organize the channel segments and elements from upstream to downstream with the channel inflow element being the first element in the file.
- The key to accurate channel routing is to balance the relationship between the slope, flow area and roughness. Channel routing is usually more stable if the natural cross section routing routine is used. Use at least 10 stations to define a cross section.
- Channel elements that are contiguous but do not share discharge (e.g. parallel channels) must be identified with the NOFLOC variables. List each pair of non-sharing contiguous channel elements only once. Review the CONFLUENCE.OUT file to make sure that you have identified all the appropriate NOFLOC's. If you miss some NOFLOC's, volume conservation may not be observed.
- If you have channel elements that will not share flow with the floodplain (either overbank or return flow), set the NOEXCHANGE parameter for the channel element. For example, closed concrete culverts which should receive no floodplain inflow to the channel.

- To improve the timing of the floodwave progression through the system, a depth variable roughness ROUGHADJ can be assigned on a reach basis. The channel roughness should be assigned for the bankfull discharge condition. Assigning ROUGHADJ will result in an increase in n-value with a decrease in flow depth during the flood simulation.

Channel Data Dependencies

It is important to pay attention to variable dependency when simulating channel flow. To simulate channel flow, complete the CHAN.DAT and XSEC.DAT files and set ICHANNEL = 1 in CONT.DAT. If you turn the ICHANNEL switch on in CONT.DAT you must consider revising other variables including:

CONT.DAT:

Assign NOPRTC = 0, 1 or 2 for addition channel output data.

INFLOW.DAT:

Set IDEPLT = grid element with a channel inflow hydrograph to plot a channel inflow hydrograph on the screen at runtime. Assign IFC = "C" for channel inflow hydrographs.

OUTFLOW.DAT:

Assign KOUT for channel outflow nodes and add the variables in Line 2 in OUTFLOW.DAT

Channel Output

Channel output can be reviewed in several ways. The channel output data is written to a series of ASCII output files including: BASE.OUT, HYCHAN.OUT, CHANMAX.OUT, DEPCH.OUT and others. The HYDROG program will display a plot of the hydrograph for each channel element. It also has a routine to review average hydraulic conditions (flow area, bed shear stress, hydraulic radius, velocity, etc.) in a channel reach covering several channel elements that user can select in the HYDROG program. The PROFILES program can be applied to review the water surface profile, spatial variation in peak discharge, mobile bed profiles, water surface in each cross section, or the cross section geometry changes associated with scour and deposition. Finally MAXPLOT and MAPPER will graphically define the relationship between channel and floodplain volumes by mapping the inundated areas.

Notes:

Modeling Guidelines

Some Basic Data File Checks

Grid System

- The grid system should begin with grid element #1 and have no missing element numbers.
- There should be no dangling grid elements connected only by a diagonal.
- If you add elements to the grid system after the model has been built, the FPLAIN.DAT and CADPTS.DAT files will have to be edited.

Channel Flow

- The channel should be organized from upstream to downstream in CHAN.DAT and should be continuous.
- At a channel confluence, the downstream main channel grid element must be lower in elevation than the confluence element.
- Eliminate channel elements with a channel length (XLEN) less than 50% of the grid element side width. Connect the channel elements across the diagonal instead.
- Create a positive bed slope at channel inflow and outflow nodes.

Inflow/Outflow Nodes

- Inflow and outflow nodes should not have other components such as hydraulic structures, streets, ARF's, etc.

- Outflow nodes should not be doubled up. Outflow nodes should have upstream floodplain elements as neighbors. Use only one line of outflow nodes.
- Create separate input hydrograph file for clear water or mudflow simulations. The sediment concentration assigned to the discretized hydrograph has to be removed for a water flood simulation. Create one file INFLOWW.DAT for water and INFLOWM.DAT mudflow, then copy the appropriate file to INFLOW.DAT before running the model.

Data Errors

Data input errors may result in the automatic termination of a simulation run along with an error message. The error message will report a 'Unit' number that is associated with the data input file that contains an error. Referring to these numbers may help you to debug input data errors.

Unit No.	File Name	Unit No.	File Name
9	TOLER.DAT	39	SED.DAT
10	CADPTS.DAT	50	OUTFLOW.DAT
30	CONT.DAT	52	STREET.DAT
31	FPLAIN.DAT	57	LEVEE.DAT
32	RAIN.DAT	68	HYDROSTRUC.DAT
33	INFIL.DAT	85	XSEC.DAT
34	INFLOW.DAT	89	RAINCELL.DAT
36	CHAN.DAT	119	CHANBANK.DAT
37	ARF.DAT	120	FPXSEC.DAT
38	MULT.DAT	180	WSTIME.DAT

A more complete list of file unit numbers can be found in the Data Input Manual.

Troubleshooting: Is the flood simulation running OK?

There are several indicators to help you identify modeling problems. The most important one is volume conservation. The FLO-2D results should be reviewed for the following: volume conservation, surging, timestep decrements, and roughness adjustments with limiting Froude numbers.

Volume Conservation

Any hydraulics model that does not report on volume

conservation should be suspected of generating or losing volume. A review of the SUMMARY.OUT file will identify any volume conservation problems. This file will display the time when the volume conservation error began to appear during the simulation. Typically a volume conservation error greater 0.001 percent is an indication that the model could be improved. The file CHVOLUME.OUT will indicate if the volume conservation error occurred in the channel routing instead of the overland flow component. Components should be switched 'off' and the model simulation run again until the volume conservation problem disappears. This will identify which component is causing the difficulty. Some volume conservation problems may be eliminated by slowing the model down (decreasing the timesteps) using the stability criteria. Most volume conservation problems are an indication of data errors.

Surging

It is possible for volume to be conserved during a flood simulation and still have numerical surging. Numerical surging is the result of a mismatch between flow area, slope and roughness. It can cause an over-steepening of the floodwave identified by spikes in the output hydrographs. Channel surging can be identified by discharge spikes in the CHANMAX.OUT file or in the HYDROG program plotted hydrographs. Predicted high maximum velocities indicate surging. To identify floodplain surging, review the maximum velocities in

the MAXPLOT or Mapper post-processor program. You can also review the VELTIMC.OUT (channel) or VELTIMFP.OUT (floodplain) files for unreasonable maximum velocities. Surging can be reduced or eliminated by adjusting (lowering) the stability criteria (DEPTOLFP or WAVEMAX in TOLER.DAT) thus decreasing the timesteps. If the decreasing the timesteps fails to eliminate the surging, then individual grid element topography, slope or roughness should be adjusted. This can be accomplished in the GDS for floodplain flow. For channel flow, the PROFILES program can be used to make adjustments. Increasing the flow roughness will generally reduce or eliminate flow surging. For channel surging, abrupt transitions in flow areas between contiguous channel elements should be avoided. Setting a lower limiting Froude number for a channel reach may also help to identify the problem.

Sticky Grid Elements

When the flood simulation is running slowly, the TIME.OUT file can be reviewed to determine the grid elements ('sticky') that are causing the most timestep decreases. TIME.OUT lists the top twenty floodplain, channel or street elements that caused the model to slow down. The file also lists whether the timestep decreases occur with the percent change in depth, Courant criteria or dynamic wave stability criteria. Adjustments can be made in the stability criteria to more equably distribute the timestep decreases. The model is designed to advance and decre-

ment timesteps, so there have to be grid elements listed in the TIME.OUT file. If one or two grid elements have significantly more timestep decreases than the other elements listed in the file, the attributes of the 'sticky' grid elements such as topography, slope or roughness should be adjusted. The goal is to make the model run as fast as possible while still avoiding numerical surging.

If a floodplain element is causing most of the timestep decreases, check the SURFAREA.OUT file to determine how much surface area is left in the floodplain element for flood storage. If the floodplain element contains a channel bank, there may be very little surface area left for flood storage. This will cause the model run slowly with exchanges the flow between the channel and floodplain. To fix this problem:

- Remove other components from the channel bank element including streets or ARF values.
- Shorten the channel length (XLEN in CHAN. DAT). This will increase the surface area in the channel bank floodplain elements.
- Decrease the channel cross section width in the PROFILES program.

Limiting Froude Numbers

There is a unique relationship between floodwave celerity and average flow velocity described by the Froude number that should not be violated during the numeri-

cal routing. This physical relationship between the kinematic and gravitation forces involves the slope, flow area and flow resistance. To use the limiting Froude number, estimate a reasonable maximum Froude number for your flood simulation and assign the value to either FROUDL (floodplain), FROUDC (channels) or STRFNO (streets) variables. When the computed Froude number exceeds the limiting Froude number, the n-value is increased by a small value (~ 0.001) for the next timestep. This change in grid element n-value helps to create a better match between the slope, flow area and n-value during the simulation. When the limiting Froude number is no longer exceeded, the n-value is gradually decreased to the original value. The changes in the n-values during the simulation are reported in the ROUGH.OUT file. For the next FLO-2D simulation, the n-value adjustments can be made to grid element using the maximum n-values reported in ROUGH.OUT. The maximum n-values are also reported in FPLAIN.RGH, CHAN.RGH and STREET.RGH files that are created at the end of a simulation. These (*.RGH) files can then be renamed to FLO-2D data input files (*.DAT) for the next flood simulation (e.g. FPLAIN.RGH = FPLAIN.DAT).

Reviewing and interpreting the results

FLO-2D results include the maximum area of inundation (maximum flow depth), temporal and spatial hydraulic results, channel or floodplain cross section hydrographs, peak discharge and other hydraulic output.

Either the MAXPLOT or the MAPPER programs can used to graphically review the model output. The flow depth results can be plotted as either line contours or shaded contours in MAPPER.

The FLO-2D flood simulation can be terminated at any time during the run by clicking on Exit on the window menu bar or by clicking the close button in the upper right hand corner of the window. The simulation will terminate after the current timestep is completed and the output files will be generated and saved. This enables the user to recognize if the flood simulation is running poorly (e.g. too slow or not conserving volume) and stop the simulation without losing the opportunity to review the output data. It is also possible to restart the flood simulation from the point that the model was terminated.

Make some adjustments

The following adjustments to the data files may improve the simulation and speed up the model:

Spatial Variation of n-values

The most common cause of numerical surging is underestimated n-values. Typical n-values represent steady, uniform flow. The spatial variation of n-values will affect the floodwave progression (travel time) and reduce surging, but may not significantly impact the area of inundation (especially for longer flood durations). Con-

centrate on the project area when adjusting n-values and review TIME.OUT and ROUGH.OUT to complete the n-value revisions.

Overland Flow Manning's n Roughness Values	
Surface	n-value
Dense turf	0.17 - 0.80
Bermuda and dense grass, dense vegetation	0.17 - 0.48
Shrubs and forest litter, pasture	0.30 - 0.40
Average grass cover	0.20 - 0.40
Poor grass cover on rough surface	0.20 - 0.30
Short prairie grass	0.10 - 0.20
Sparse vegetation	0.05 - 0.13
Sparse rangeland with debris	
0% cover	0.09 - 0.34
20 % cover	0.05 - 0.25
Plowed or tilled fields	
Fallow - no residue	0.008 - 0.012
Conventional tillage	0.06 - 0.22
Chisel plow	0.06 - 0.16
Fall disking	0.30 - 0.50
No till - no residue	0.04 - 0.10
No till (20 - 40% residue cover)	0.07 - 0.17
No till (60 - 100% residue cover)	0.17 - 0.47
Open ground with debris	0.10 - 0.20
Shallow Flow on asphalt or concrete (0.25" to 1.0")	0.10 - 0.15
Fallow fields	0.08 - 0.12
Open ground, no debris	0.04 - 0.10
Asphalt or concrete	0.02 - 0.05

Edit Topography

The interpolation of DTM points to assign elevations to grid elements is not perfect even when the GDS filters are applied. It may be necessary to adjust some floodplain grid element elevations when you review the results. MAXPLOT and Mapper can be used to quickly locate grid elements with unreasonable flow depths that may constitute inappropriate depressions. Floodplain depressions can sometimes occur along a river channel if too many DTM points located in the channel topography.

Floodplain Surface Area Reduction

The distribution of flood storage on the grid system can be influenced by assigning area reduction factors (ARF's). For large flood events, the assignment of individual grid element ARF values will usually have minor impact on the area of inundation. For local flooding detail, individual grid element ARF assignment may be justified.

Channel Cross Section Adjustments

Typically a surveyed cross section will represent five to ten channel elements. Selecting a cross section to represent transitions between wide and narrow cross sections requires engineering judgment. Use the PROFILES program to interpolate the transition between surveyed

cross sections.

Channel Slope Adjustments

Adverse channel slopes can be simulated by FLO-2D. Smoothing out an irregular slope condition over several channel elements to represent reach average slope conditions may speed up the simulation. Cross sections with scour holes can result in local adverse slopes that misrepresent the average reach conditions. Review the channel slope in PROFILES.

Street Flow

Streets generally convey only a small portion of the flood volume, but may be important for flood distribution to remote areas of the grid system. Streets are important to flood delineation in urban areas. High street velocities may cause numerical surging and slow the simulation down. Assign reasonable limiting street Froude numbers to adjust the street n-values.

A few important things to consider

Some modeling tips for working with unconfined water flood simulations are presented.

Rainfall and Infiltration on Alluvial Fans

Alluvial fan surfaces can be as large as the upstream watershed. Fan rainfall can contribute a volume of water on the same order of magnitude as the inflow flood

hydrograph at the fan apex. Infiltration losses can also significantly effect floodwave attenuation. Infiltration losses should be calibrated to the watershed percent loss by adjusting the hydraulic conductivity. Spatial variability of the hydraulic conductivity can be assigned with the GDS program.

Sediment Bulking of Flood Hydrographs

For a mudflow alluvial fan simulation, sediment concentration can be adjusted in the INFLOW.DAT file. For desert alluvial fans, sediment concentrations in flood events can reach 15% by volume. For concentrations less than 20% by volume, the flow will behave like a water flood. The primary effect of increasing the sediment concentration is to bulk the flow volume. Do not invoke the mudflow component (MUD = 1 in CONT.DAT) unless sediment concentrations greater than 20% by volume are expected. Use the XCONC factor (CONT.DAT) to bulk desert alluvial fan flows by 10% to 15 % by volume. Do not set both the MUD and ISED switches to "on" in the CONT.DAT file in the same simulation.

Model Calibration and Replication of Flood Event

Estimating flood hydrology (both rainfall and flood hydrographs) can represent a significant departure from reality when replicating historical floods. When attempting to match measured flood stages, high water marks or channel discharges, focus first on obtaining a reasonable

estimate of the flood volume, then concentrate on the model details such as n-values, ARF's and street flow. Remember that flood volume is more important than peak discharge for a flood routing simulation.

How can I improve model speed and stability?

The FLO-2D model uses dynamic wave stability criteria (WAVEMAX) as one option to control the magnitude of the computational timestep. The WAVEMAX coefficient is applied to the overland, street and channel components. The dynamic wave stability criteria is essentially an extension of the Courant criteria used for diffusive wave flood routing and includes a slope term and specific discharge in place of the velocity and wave celerity. The purpose of WAVEMAX is to provide more strict control of the timestep when analyzing complex and rapidly varying flow in channels such as channel transitions, confluences and split flow reaches. Several changes can be made to the application of the dynamic wave stability criteria to increase model speed.

For most overland flow applications including those involving overbank river flooding, the Courant stability criteria is sufficient. This numerical stability criteria is hard-wired in the model using a Courant coefficient = 1.0. The dynamic wave stability criteria for more complex flows relates the discretized time and space system to the movement of the floodwave in terms of slope and specific discharge for individual grid elements:

$$\Delta t < \zeta S_o \Delta x^2 / q_o$$

where: Δt = limiting computation timestep
ζ = dynamic wave stability coefficient (WAVEMAX)
S_o = bed slope
Δx = grid element width
q_o = specific discharge

For a floodplain, channel or street grid element, the WAVEMAX value increments and decrements based on whether the computed timestep is exceeded.

How does WAVEMAX speed up the FLO-2D model?

There are three options for applying the overland flow dynamic wave stability criteria:

1. Dynamic wave stability criteria varies the model timestep when WAVEMAX is set within the range 0.10 to 1.00 (typical value = 0.25). This approach makes the model run slower, but is more numerically stable.

2. The floodplain Manning's n-values are incremented when the dynamic wave stability criteria is exceeded, but the timestep is not decreased. Assign WAVE-MAX as a negative number using same range of values -0.10 to -1.00 (typical value = -0.25). The runtime changes in the n-value will be written to the ROUGH.OUT file. The variation of the n-values occur according to the following relationships:

i. $n = n + 0.0006616\ e^{(-10.9\ n)}$ when the limiting timestep is exceeded.

ii. $n = n - 0.00005$ when the limiting timestep is not exceeded.

This approach uses WAVEMAX to identify problem elements, but then also uses increased n-values to resolve the mismatches between the discharge, slope, roughness and flow area instead of reducing the time step.

3. The dynamic wave numerical stability criteria are turned off when WAVEMAX is assigned a value of 100 or more. Assign WAVEMAX = WAVEMAX + 100. The timesteps are varied only by the change in depth (DEPTOL variable) or the Courant stability criteria.

   The guidelines for applying these options are as follows:

   - Run the model with WAVEMAX = -0.25 and an appropriate limiting Froude number (e.g. FROUDL = 0.9 subcritical flow on an alluvial surface). This will calibrate the model n-values for reasonable Froude numbers and for the movement of the floodwave using the dynamic wave stability criteria.

- Review the maximum velocities in MAXPLOT or MAPPER to determine the location of any inappropriate high velocities related to numerical surging and increase the n-values of all the grid elements in the vicinity.
- Review the n-values in ROUGH.OUT. Make n-value adjustments in FPLAIN.RGH for any high n-values in ROUGH.OUT. Also make roughness adjustments for any observed high maximum velocities, then replace FPLAIN.DAT with FPLAIN.RGH.
- Run the simulation and continue to replace FPLAIN.DAT until ROUGH.OUT is essentially empty. A few incremental n-values changes in ROUGH.OUT will not affect the simulation. You can also make adjustments to WAVEMAX and FROUDL to decrease the number of required n-value adjustments.
- Set WAVEMAX = WAVEMAX + 100 and run the model again. The model will run faster. Check the maximum velocities for any inappropriate high velocities and make n-value adjustments. If the model has numerical surging, set WAVEMAX = 0.25 and run the model. The model will run slower, but should eliminate the numerical surging.

What are the results of applying different options?

You should notice an increase in model speed when se-

lecting option 2 or 3 above. Choosing option 2 has the effect of improving the spatial distribution of reasonable n-values and reducing the number of choke points associated with numerical instability. Varying n-values will help to control the grid element discharge flux to accommodate the movement of the floodwave. Choosing option 3 essentially results in the overland flow timestep being controlled only by the Courant criteria and the model runs much faster. Option 3 should be applied to the final production simulations. The result of following this procedure will be:

- A calibrated model for overland roughness values in which the discretized numerical system in both time and space will better simulate the movement of the floodwave.
- The floodwave speed will be bound by a reasonable limiting Froude number.
- The model will run fast without numerical surging or oversteepening of the floodwave.

Notes:

Making Flood Maps

Mapper and Mapper_NET create a diverse array of high resolution graphical plots including flood hazard maps including:

- Ground surface elevation
- Maximum water surface elevation
- Maximum depth (area of inundation)
- Maximum velocity
- Final depth
- Final velocity
- Specific Energy
- Impact Pressure
- Static Pressure
- Time-to-Peak Discharge (dam and levee break)
- Time-to-One Foot (dam and levee break)
- Time-to-Two Foot (dam and levee break)
- Temporally variable depth and velocity
- Flow depth over DTM points
- Flood hazard maps
- Shape files
- DFIRM's

Some of the maps can be generated for floodplain, channel and street flow. Combined channel and floodplain maximum depth plots can also be generated. Most of the maps can be displayed as either grid element plots, line contour maps, and shaded contour maps. Shape files for importing results to GIS are automatically

generated for most of the Mapper plots. Some guidelines to consider when developing flood maps are:

- The flood hazard map resolution is only as good as that of the topographic data base;
- Use background aerial images to enhance the maps;
- Contour line width and shaded contour splash over floodplain features can affect resolution appearance;
- Computing and plotting flow depths over the DTM points improves the inundation map resolution;
- Map resolution controls include contour intervals, deleting the lowest contour, and contour smoothing;
- Mapper_NET has advanced mapping features to improve map resolution including breaklines.

All CADD and GIS programs have to accommodate topographic data base resolution and contour splash (e.g. flood contours that cover levees, buildings, bluffs or other features). Map resolution is a function of the point density (grid element spacing), contour line width and plotting algorithm. Mapper and Mapper_NET have options to address these resolution issues including:

Maximum flood depth computation over DTM points.

By importing the DTM ground elevation points into Mapper and subtracting the ground elevation from the FLO-2D predicted maximum grid element water surface elevation, flow depths are computed for every DTM point. Plotting the DTM point flow depth shaded con-

tours instead grid element shaded contours will greatly enhance the map resolution. A file (FLO2DGIS.OUT) of these DTM point flow depths can also be created for importing to GIS.

Shape file generation.

Mapper and Mapper_NET automatically create shape files in the project folder that can be imported to GIS or CADD programs for further editing.

Breaklines and other map controls.

Mapper_NET is integrated with ArcGIS Runtime Engine Controls. It has a number of contour enhancements including a vast array of color combinations. More importantly, there are breakline options to limit contours from crossing topographic features.

Creating high resolution flood maps may require several rounds of review and adjustment of your mapping controls. The final map product, however, will only be as good as the topographic data base from which it was prepared.

www.flo-2d.com

FLO-2D Porting Document (Version 2009)

This document will describe the data file changes necessary to update older model data files to version 2009. Porting descriptions are listed from version 2004 to version 2006, version 2006 to 2007, and version 2007 to 2009.

Porting from Version 2007 to Version 2009

There are very few data input changes to convert Version 2007 data files to Version 2009. Except for adding groundwater model MODFLOW switch in CONT.DAT and the new CHANBANK.DAT file, all the changes involve optional data or components. The optional data file revisions do not require any 2007 data files modifications for the FLO-2D model to run. To convert v2007 data files to v2009 simply open the v2007 data files in the GDS and save them. Nothing else is necessary. Another approach is to simply review the data files with the Data Input Manual description and make any necessary changes.

CONT.DAT

Surface water - groundwater interaction can now be simulated through the integration at runtime with the MODFLOW model. The new switch parameter to initiate the groundwater model is IMODFLOW at the end of line 3 after the ISED parameter. For a default of no groundwater simulation set IMODFLOW = 0.

CHAN.DAT

The critical model revision that must be addressed is the elimination of the IQDIR channel extension direction from the CHAN.DAT file. The data input change is automated in the GDS and FLOENVIR. The required data revision is as follows:

Delete IQDIR from Line 2. This can be done manually. It is only necessary, however, to open and save the CHAN.DAT in the GDS or FLOENVIR and save the files and the IQDIR variable will be deleted and a new data file CHANBANK.DAT will be generated. This new file contains the left and right bank channel elements and it is automatically generated when the channel is saved in the GDS and FLOENVIR. After the CHANBANK.DAT is generated, it may be necessary to make some minor adjustments to the automated selection of the right banks. Again this can be done graphically in the GDS or FLOENVIR.

Optional Data File Revisions

The Courant Number can now be assigned in TOLER.DAT as line 2. See the PDF handout entitled "Courant Number Instructions".

Levee fragility curves can be assigned in the BREACH.DAT file and the global and individual levee curve assignment to the levee elements is accomplished in the LEVEE.DAT file. The new levee fragility curve data is appended to the each file. Refer to the respective data file descriptions to input the fragility curve parameters.

For surface water/groundwater interaction at runtime, review the required MODFLOW parameters in the MODFLOW.DAT file in the Data Input Manual.

Additional Output Files

Three new output files are now generated:

FPINFILTRATION.OUT *contains the total cumulative infiltration by grid element.*

VELTIMC.OUT and VELTIMFP.OUT lists maximum velocity and time of occurrence in descending order for the channel and floodplain respectively.

Porting from Version 2006 to Version 2007

Convert Version 2006 FLO-2D data files to the Version 2007. These data file revisions were made to simplify the data input and to accommodate new variables. No conversion program was created to perform the data file revision because they are so simple. Edit the data files as follows:

CONT.DAT

Delete INPLOT. This variable is now assigned automatically.

CHAN.DAT

Delete ICHDEP from Line 1. This variable is now assigned automatically. Please note that more than one line 1 may appear in the file.

LEVEE.DAT

Add ILEVFAIL to Line 1. This is new switch to identify the levee failure mode. Set ILEVFAIL = 0 for no levee failure. Set ILEVFAIL = 1 for prescribed level failure rates of breach opening. Set ILEVFAIL = 2 for initiating the levee or dam breach failure routine.

The following data files changes are optional and are not required to make a Version 2006 data set model run with the Version 2007 model.

CONT.DAT

Revised AMANN to accept -99 to turn the depth variable n-values for the floodplain.

INFLOW.DAT

Line 4 has been added to enable automatic assignment of water surface elevation for reservoirs or ponded areas. Line 4 includes the character 'R', a grid element within the reservoir area IRESGRID and the water surface elevation RESERVOIREL of the FLO-2D simulation. See INFLOW.DAT file description in the Data Input Manual.

TOLER.DAT

Revised WAVEMAX values to include a negative number that will turn off the dynamic wave stability criteria. See the TOLER.DAT file description for a complete discussion of the new stability criteria options.

The SCS curve number routine for infiltration has been added to the FLO-2D, but will not be functional until July 1, 2007.

INFIL.DAT

Add INFMETHOD in line 1. INFMETHOD is a new switch to identify the infiltration equation. Set INFMETHOD = 1 to simulate infiltration with the Green-Ampt method. Set INFMETHOD = 2 to simulate infiltration with the SCS curve number method. If you set INFMETHOD = 1, no further changes are necessary to the version 2006 INFIL data file. If you use the SCS method, you will have add a new line 4 with the SCSCNALL and new line 5 with INFILCHAR = 'S', INFGRID(N), SCSCN(N); the individual grid element SCS values.

Porting from Version 2004 to Version 2006

A program called CONVERT.EXE can be downloaded from the website www.flo-2d.com to automatically port from version 2004 to version 2006. Simply download the program and copy it into a folder that contains the 2004 FLO-2D data files (*.DAT). Double click CONVERT.EXE to run the program and it will find and convert the data files in the folder. Once the data files have been updated to Version 2006, the preceding porting instructions can be used to update the data files to Version 2009.

Porting from Model Earlier than Version 2004

Any FLO-2D version data files prior to Version 2004 have to be updated manually by reviewing the data file descriptions in the data input manual. Open the old version data files in WordPad© or any ASCII editor and adjust the data parameter format. For large or complex models, it may be cost effective to have the FLO-2D staff convert the data files to 2009 for an hourly fee. For more information, email us at contact@flo-2d.com.

FLO-2D Limiting Froude Number Application Guidelines

Using the Froude number in flood routing models is an important to the both the understanding of the floodwave movement and the numerical stability of the model. It is even more important when considering mobile bed channels such as alluvial fans or high bedload rivers. The dimensionless Froude number F_r is given by:

$$F_r = \alpha^{0.5}\, V/(gd)^{0.5}$$

where V = depth averaged flow velocity, d = depth of flow above the thalweg, and α = kinetic energy correction factor (involving the fluid density and specific weight). Typically α is assumed to be 1. The Froude number helps to define the influence of gravity on the flow pattern. A low wave will propagate in free surface flow (open channel) depending only on the gravitational acceleration and the flow depth. The movement (speed) of the shallow wave, known as the wave celerity $c = \sqrt{gd}$, is related to average flow velocity through the Froude number. By accepting reasonable limitations of the overland (or channel) flow velocity and floodwave movement, the Froude number can be used to further define the relationship between the velocity and flow depth.

For essentially steady and uniform flow, the Manning's n value is defined would be defined by:

$$n = (0.262/F_r)\, d^{0.17}\, S_o^{0.5}$$

indicating that the flow roughness is inversely proportional the Froude number. By assuming a reasonable limiting Froude number, the n value can be estimated from the normal depth and slope for a given flow discharge.

In the FLO-2D model, the suggested n-value is based on either bankfull discharge for channel flow or 1 meter (3 ft) flow depth for overland flow (roughness is fully submerged). Suggested typical limiting Froude numbers are:

	Flat or Mild Slope (large rivers and floodplains)	Steep Slope (alluvial fans and watersheds)
Channels	0.4 – 0.6	0.7 - 1.05
Overland flow	0.5 – 0.8	0.7 – 0.95
Streets	0.9 – 1.2	1.1 – 1.5

Similar values are also reported in the CVFED FLO-2D Application Guide. If the limiting Froude number is exceeded, the grid element n-value increases by 0.001 for the next timestep. When limiting F_r is no longer exceeded, the n-value decreases by 0.0005 if it's greater than the original n-value. The changes in n-value reported in ROUGH.OUT, FPLAIN.RGH, CHAN.RGH and STREET.RGH files. The use of limiting Froude number in the FLO-2D model is documented in the FLO-2D Pocket Guide, the FLO-2D Reference Manual, and the CVFED FLO-2D Application Guide.

Additional considerations:

1. The maximum n-values for discretized models will be greater than typical n-values for HEC-RAS cross sections (both channel and overbank areas). This is because of the unsteady and non-uniform flow contribution between elements and the flow not being parallel to the cross section.
2. For flows over mobile bed conditions (supply unlimited), critical flows are approached asymptotically (Grant, 1997). A relatively steep slope is required for flow with sediment transport to achieve critical flow because the flow hydraulics oscillate. For $F_r > 1$, flow instability leads to rapid energy dissipation and bed erosion. Flow is forced to stay around critical by incipient motion thresholds. The equilibrium sand bed morphology tends to minimize the Froude number (Jia, 1990).
3. The limiting Froude number for mobile bed conditions can be approximated by (Grant, 1997):

$$F_r = 3.85\ S^{0.33} \quad \text{gravel bed } (\tau^*_{cr} = 0.03)$$
$$F_r = 5.18\ S^{0.11} \quad \text{sand bed } (\tau^*_{cr} = 0.06)$$

4. Roughness n-values include many factors: $n = n_1 + n_2 + n_3 + n_4 + \ldots$such as friction drag, vegetation, expansion/contraction, bed forms, flow in bends, unsteady and non-uniform flow

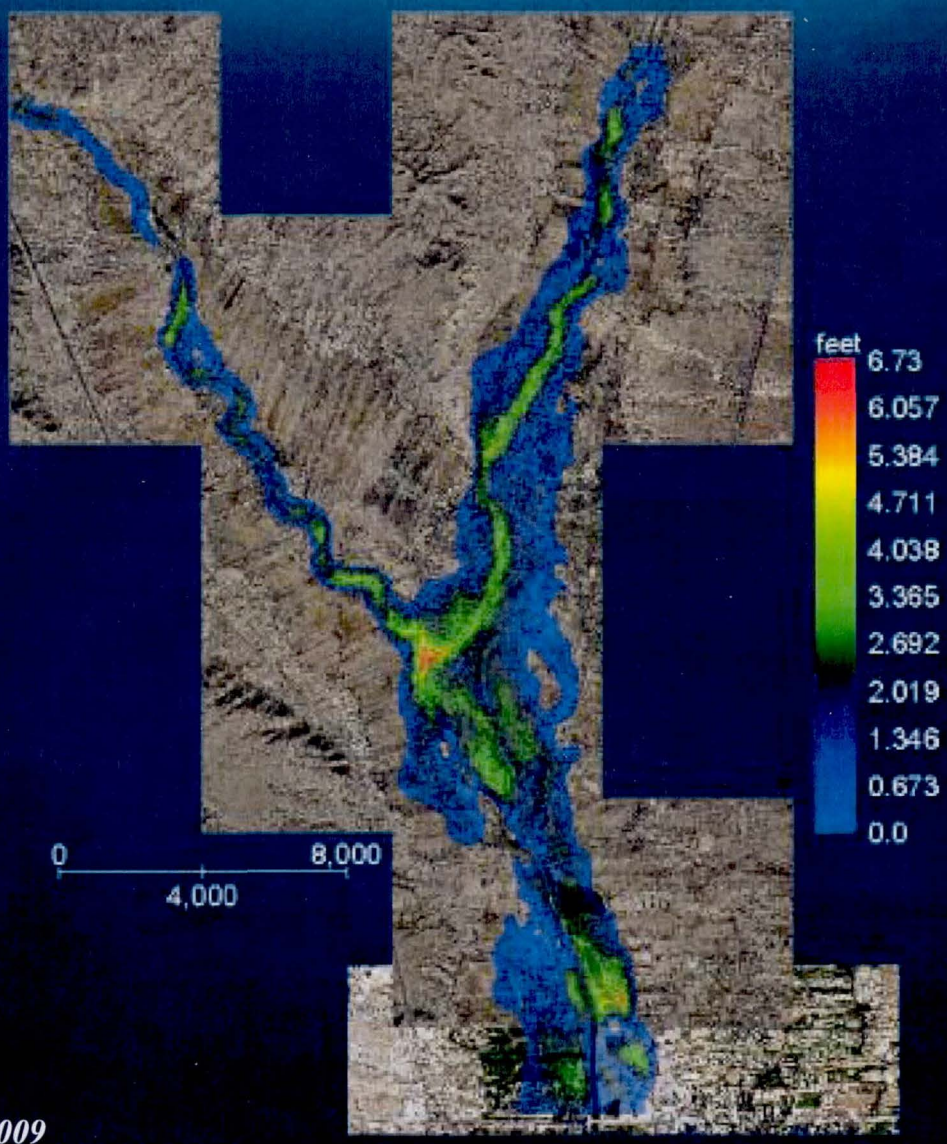
References:

Grant, G.E., 1997. "Critical flow constrains flow hydraulics in mobile bed streams: A new hypothesis," Water Resources Research, V. 33, No. 2, pp. 349-358.

Jia, Y., 1990. "Minimum Froude number and the equilibrium of alluvial sand rivers," Earth Surface, Processes, and Landforms, 15, pp. 199-209.

FLO-2D

Reference Manual



Version 2009

A few comments on modeling free surface flows...

With faster computers and higher resolution digital terrain models, flood routing models are becoming very detailed. When adding detail to a two-dimensional flood routing model, a number of factors should be considered including flood hydrology accuracy, topographic model resolution, spacing of the channel cross sections, and limited calibration data. As flood models become more detailed, the user should try to find a balance between model resolution, computer resources and budget.

Reliable flood hazard delineation requires a critical review of model applicability, modeling assumptions, and the available data bases. While finite difference models have become more versatile with increasing computer resources, inadequate hydrographic data bases still limit the accuracy of flood hazard delineation. Digital terrain models are becoming the foundation of high resolution mapping, but post-flood event surveys of high water marks and aerial photography of the area of inundation are either unavailable or perhaps were collected long after the flood waters have receded. Correlating the area of inundation with flood peak discharge can lead to the harsh realization that our best discharge measurements or gaging data have limited accuracy at high flows. Our modeling and mapping results may be only as good as the model calibration to post-flood data.

As flood inundation mapping advances with hydrograph routing, extensive topographic data bases, high resolution graphics, and unconfined hydraulic modeling, it may appear that flood modeling complexity is becoming overwhelming. Please take heart in the comments of Cunge et al. (1980):

> *"The modeler must resist the temptation to go back to one-dimensional schematization because of lack of data otherwise necessary for an accurate two-dimensional model calibration. If the flow pattern is truly two-dimensional, a one-dimensional schematization will be useless as a predictive tool..." "It is better to have a two-dimensional model partially calibrated in such situations than a one-dimensional one which is unable to predict unobserved events. Indeed, the latter is of very little use while the former is an approximation which may always be improved by complimentary survey."4*

As a final word, please remember that all software programs has an occasional glitch. Modeling bugs are inherent part of the process of adding new routines and attempting to make the model run faster. Even when a model engine is fine tuned, adding components may introduce conflicts with older subroutines or perhaps may uncover bugs that were previously undetected. FLO-2D is no exception. Version 2007 will run faster than previous models and when comparing results with previous versions, you may note some minor differences associated with the larger computational timesteps. Generally, the Version 2007 FLO-2D results should be more accurate, but we will immediately address all questions concerns over model application, accuracy or problems. On occasion there is a project application that pushes the model to new limits. Such projects can lead to new developments that benefit all users. The modeler is encouraged to share interesting projects with us. We aspire to make the FLO-2D model a comprehensive and flexible tool.

BRIEF OVERVIEW

FLO-2D is a volume conservation flood routing model. It is a valuable tool for delineating flood hazards, regulating floodplain zoning or designing flood mitigation. The model will simulate river overbank flows, but it can also be used on unconventional flooding problems such as unconfined flows over complex alluvial fan topography and roughness, split channel flows, mud/debris flows and urban flooding. FLO-2D is on FEMA's list of approved hydraulic models for both riverine and unconfined alluvial fan flood studies.

The FLO-2D software package includes a grid developer system (GDS), a Mapper program that automates flood hazard delineation, and the FREQPLOT program to analyze flood frequency. The GDS will filter DTM points, interpolate the DTM data and assign elevations to grid elements. The MAPPER program automates flood hazard delineation. MAPPER will generate very detailed flood inundation color contour maps and shape files. It will also replay flood animations and generate flood damage and risk maps. A graphical user interface (GUI) has been developed to assist the user in preparing and editing the data files.

The FLO-2D Reference Manual is devoted to a model description, theory and components. The user is encouraged to read this manual to become familiar with the overall model attributes and equations. The Data Input Manual is subdivided into a series of data files with variable descriptions and comments. The user should consult this manual when constructing data files. Separate manuals are devoted to the application of the GDS and Mapper.

The user can keep current on FLO-2D model and processor updates, training and other modeling news at the website: www.flo-2d.com.

FLO-2D Software, Inc.
P.O. Box 66
Nutrioso, AZ 85932

Phone and FAX: (928) 339-1935
Email: contactus@flo-2d.com

TABLE OF CONTENTS

Page

BRIEF OVERVIEW ii
LIST OF FIGURES iv
LIST OF TABLES iv

I. INTRODUCTION 1
1.1 Evolution of the FLO-2D Model 1
1.2 Modeling the Hydrologic System with FLO-2D 2
1.3 Getting Started on a Project – A Brief Overview 5

II. FLO-2D MODEL THEORY 7
2.1 Governing Equations 7
2.2 Solution Algorithm - How the Model Works 8
2.3 The Importance of Volume Conservation 11

III. FLO-2D MODEL SYSTEM 14
3.1 Assumptions 14
3.2 Parameter Variability 16
3.3 Inflow and Outflow Control 18
3.4 Floodplain Cross Sections 18
3.5 Graphical User Interface 19
3.6 Grid Developer System (GDS) 19
3.7 Graphical Output Options 19
3.8 Data Output Options 20

IV. MODEL COMPONENTS 21
4.1 Model Features 21
4.2 Overland Flow 21
4.3 Channel Flow 25
4.4 Channel-Floodplain Interface 26
4.5 Limiting Froude Numbers 27
4.6 Levees 28
4.7 Levee and Dam Breach Failures 30
4.8 Hydraulic Structures 36
4.9 Street Flow 36
4.10 Floodplain Surface Storage Area Modification and Flow Obstruction 37
4.11 Rainfall and Runoff 38
4.12 Infiltration and Abstraction 39
4.13 Evaporation 42
4.14 Overland Multiple Channel Flow 42
4.15 Sediment Transport – Total Load 43
4.16 Mud and Debris Flow Simulation 47

V. FLO-2D APPLICATIONS AND METHODS 56
5.1 River Applications 56
5.2 Unconfined Overland and Alluvial Fan Floodimg 57
5.3 Model Results – What Constitutes a Successful Flood Simulation? 59

VI. FLO-2D MODEL VALIDATION 59

VII. REFERENCES 61

LIST OF FIGURES

Page

Figure 1. Physical Processes Simulated by FLO-2D 3
Figure 2. Channel – Floodplain Interface 4
Figure 3. FLO-2D Flow Chart 6
Figure 4. Discharge Flux across Grid Element Boundaries 12
Figure 5. FLO-2D Stability Criteria Flow Chart 13
Figure 6. Overland Tsunami Wave Progression in an Urban Area (Waikiki Beach, Hawaii) 18
Figure 7. Overland Flow Routing Subroutine Flow Chart 24
Figure 9. Channel Extension over Several Grid Elements 25
Figure 9. Levees are depicted in Red and the River in Blue in the GDS Program 29
Figure 10. Levee Freeboard Deficit Plot in Mapper 30
Figure 11. Example of Levee Breach Urban Flooding 30
Figure 12. Example of a Proposed Domestic Water Supply Reservoir Breach Failure 31
Figure 13. Pipe Breach Failure 32
Figure 14. Overtopping and Channel Breach Erosion 33
Figure 15. Breach Schematic Flow Chart 35
Figure 16. Streets Depicted in Green in the FLOENVIR Program. 37
Figure 17. Area and Width Reduction Factors 38
Figure 18. Flooding Replicated from NEXRAD Data near Tucson, Arizona 39
Figure 19. Sediment Transport Bed Exchange Layer 44
Figure 20. Classification of Hyperconcentrated Sediment Flows 48
Figure 21. Shear Stress as a Function of Shear Rate for Fluid Deformation Models 51
Figure 22. Dynamic Viscosity of Mudflow Samples versus Volumetric Concentration 55
Figure 23. Yield Stress of Mudflow Samples versus Volumetric Concentration 55
Figure 24. Middle Rio Grande and Rio Chama Confluence Model 57
Figure 25. Unconfined Alluvial Fan Flooding 57
Figure 26. Urban flooding with Street Flow and Building Obstruction 58
Figure 27. FLO-2D versus USGS Measured Gage Data 60

LIST OF TABLES

Table 1. Overland Flow Manning's n Roughness Values 22
Table 2. Initial Abstraction 40
Table 3. Green Ampt Infiltration - Hydraulic Conductivity and Porosity 40
Table 4. Green Ampt Infiltration - Soil Suction 41
Table 5. Green Ampt Infiltration -Volumetric Moisture Deficiency 41
Table 6. Mudflow Behavior as a Function of Sediment Concentration 49
Table 7. Resistance Parameters for Laminar Flow 52
Table 8. Yield Stress and Viscosity as a Function of Sediment Concentration 53

FLO-2D
Two Dimensional Flood Routing Model

I. INTRODUCTION

This Reference Manual discusses the physical processes of flooding. It is designed to acquaint the user with the model theory, finite difference algorithms, model components, modeling assumptions and limitations, and potential flood scenarios. A reference list is provided for further reading.

1.1 Evolution of the FLO-2D Model

The first version of the FLO-2D model was called MUDFLOW. It was initiated in 1988 to conduct a Federal Emergency Management Agency (FEMA) flood insurance study (FIS) of an urbanized alluvial fan in Colorado. FEMA had requested the investigation of flood routing models that might be suitable for simulating mudflows. The Diffusive Hydrodynamic Model (DHM) created by Hromadka and Yen (1987) distributed by the USGS was considered to be a simple finite difference model that might serve as a template to develop a more sophisticated hydraulic model for mudflows. The selection of the DHM model as a template for the MUDFLOW model was based on its availability in the public domain, its simple numerical approach and a finite difference scheme that permitted modification of the grid element attributes.

The original MUDFLOW model was only a few hundred lines of Fortran code and was limited to 250 grid elements. A six hour hydrograph took over 12 hours to run on an XT computer. After 21 years of development, the program code has grown to be in excess of 40,000 lines of code, 60 subroutines and a number of processor programs. Virtually none of the original simplistic DHM concept remains in the current FLO-2D model. FLO-2D computes overland flow in 8-directions, reports on mass conservation, utilizes a variable timestep incrementing and decrementing scheme, incorporates efficient numerical stability criteria, has unlimited array allocation (unlimited grid elements), includes graphical editing, and has output display processor programs.

FLO-2D is a physical process model that routes rainfall-runoff and flood hydrographs over unconfined flow surfaces or in channels using the dynamic wave approximation to the momentum equation. It has a number of components to simulate street flow, buildings and obstructions, sediment transport, spatially variable rainfall and infiltration, floodways and many other flooding details. Predicted flow depth and velocity between the grid elements represent average hydraulic flow conditions computed for a small timestep (on the order of seconds). Typical applications have grid elements that range from 25 ft to 500 ft on a side and the number of grid elements is unlimited.

1.2 Modeling the Hydrologic System with FLO-2D

The FLO-2D system consists of processor programs to facilitate graphical editing and mapping and components that simulation channel and floodplain detail. The Grid Developer System (GDS) generates a grid system that represents the topography as a series of small tiles. The FLO-2D model has components for rainfall, channel flow, overland flow, street flow, infiltration, levees and other physical features. The GDS and the FLOENVIR processor programs are used to spatially edit the grid system attributes. The PROFILES program edits channel slope and cross section shape. Flood routing results can be viewed graphically in the MAXPLOT, MAPPER and HYDROG (plot hydrograph) programs.

FLO-2D is an effective tool for delineating flood hazards or designing flood mitigation. The model utility is discovered through its application to diverse flooding problems. Starting with a basic overland flood scenario, details can added to the simulation by turning on or off switches for the various components shown in Figure 1. Multiple flood hydrographs can be introduced to the system either as a floodplain or channel inflow. As the floodwave moves over the floodplain or down channels or streets, flow over adverse slopes, floodwave attenuation, ponding and backwater effects can be simulated. In urban areas, buildings and flow obstructions can be simulated to account for the loss of storage and redirection of the flow path. The levee component can be used to test mitigation alternatives.

Channel flow is one-dimensional with the channel geometry represented by either by natural, rectangular or trapezoidal cross sections. Street flow is modeled as a rectangular channel. Overland flow is modeled two-dimensionally as either sheet flow or flow in multiple channels (rills and gullies). Channel overbank flow is computed when the channel capacity is exceeded. An interface routine calculates the channel to floodplain flow exchange including return flow to the channel. Similarly, the interface routine also calculates flow exchange between the streets and overland areas within a grid element (Figure 2). Once the flow overtops the channel, it will disperse to other overland grid elements based on topography, roughness and obstructions. For flood projects with specific requirements, there are several unique components such as mud and debris flow routing, sediment transport, a floodway option, open water surface evaporation and others.

The user is encouraged to apply these components while understanding the contribution of each component to the overall flood distribution. It is important to assess the level of detail required on a given project. FLO-2D users have a tendency to put more detail into their models than is necessary for a large flood event. Preparation of channel flow, street flow, buildings and flow obstructions data files can be time consuming and should be tailored to meet the project needs. The desired accuracy of predicted water surface elevations should be consistent with the resolution of the mapping, survey and hydrologic data bases. Simulating large floods requires less detail than shallow flood or mitigation design models. Grid element sizes ranging from 25 ft (8 m) to 500 ft (150 m) is practically for most flood inundation projects.

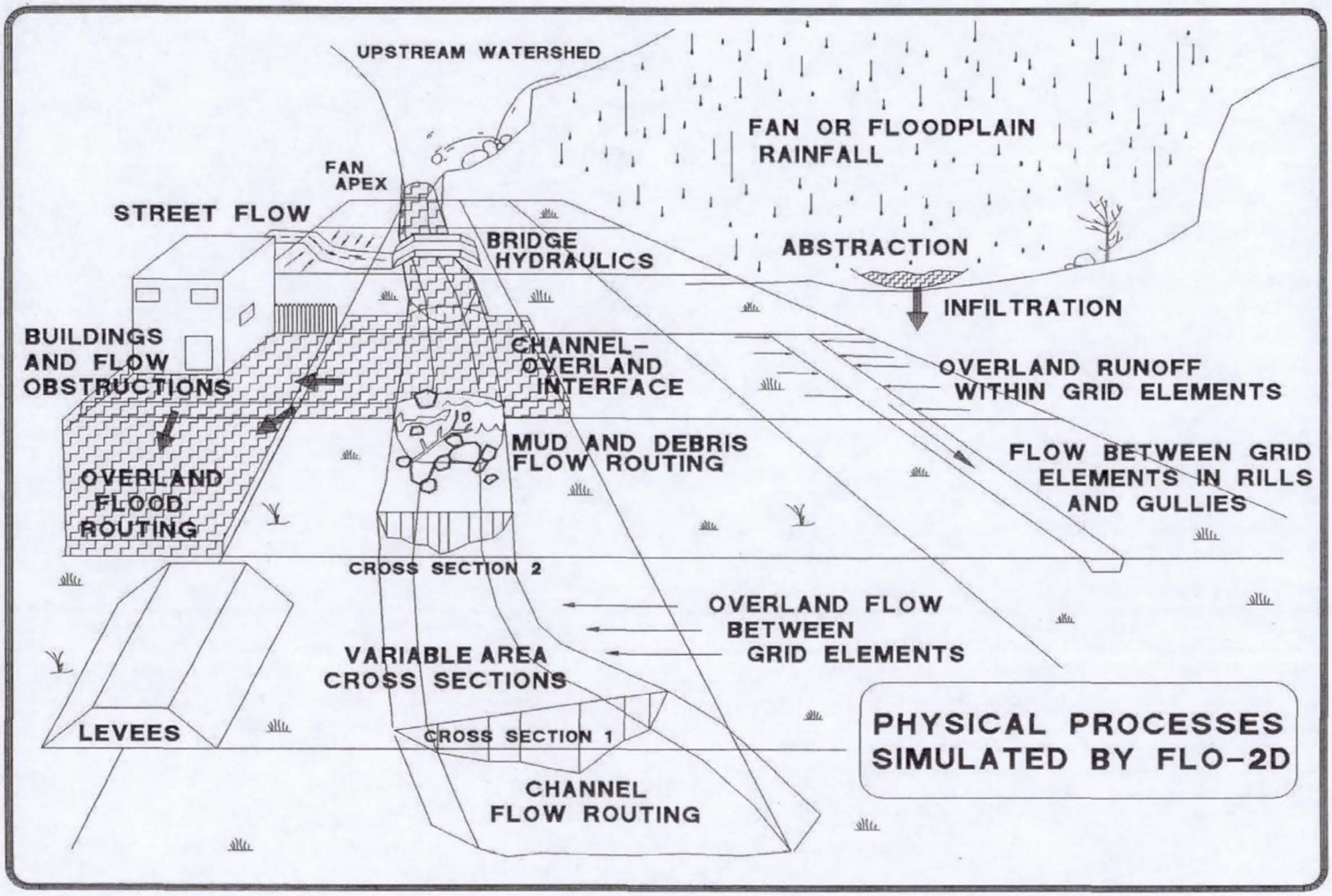


Figure 1. Physical Processes Simulated by FLO-2D

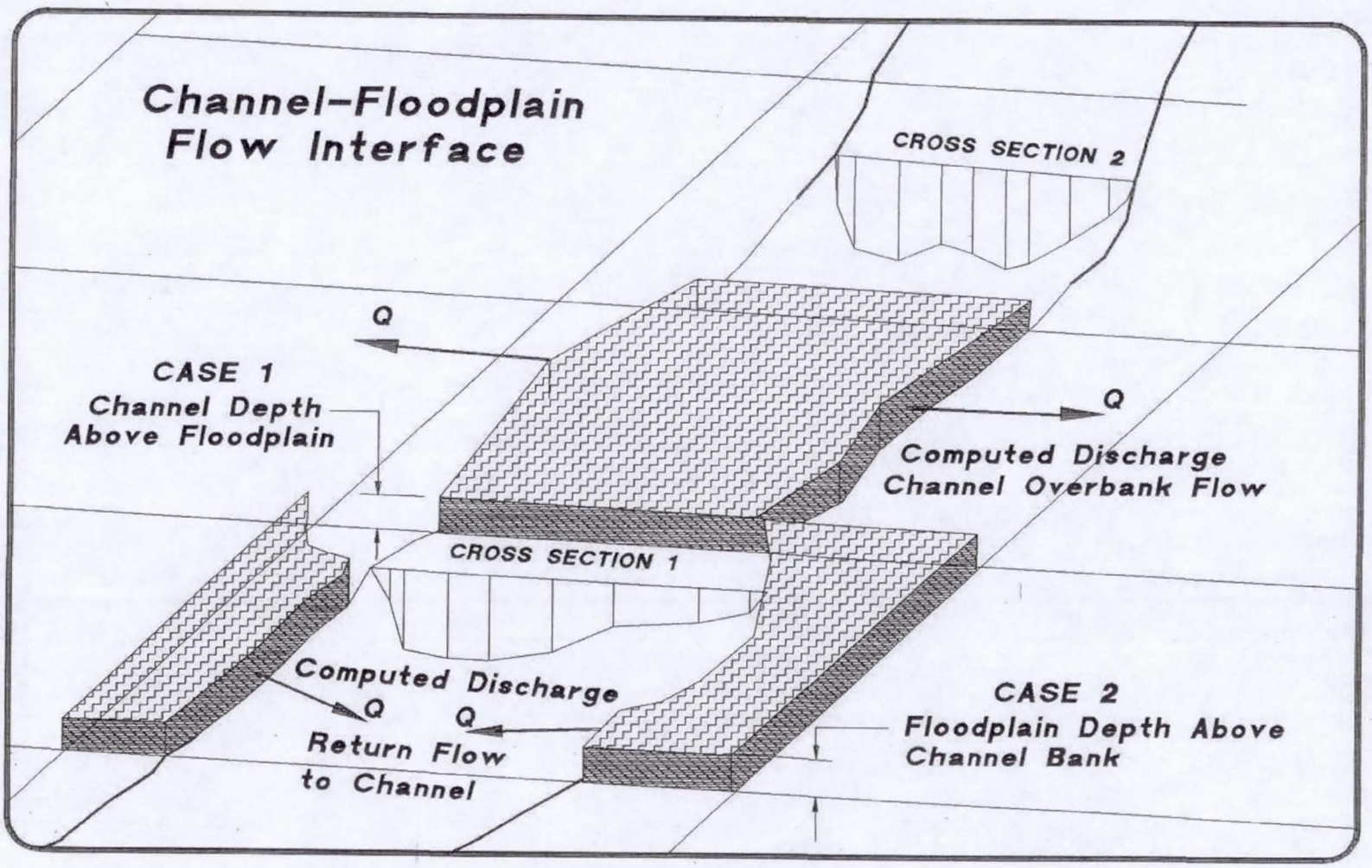


Figure 2. Channel – Floodplain Interface

1.3 Getting Started on a Project – A Brief Overview

There are two important steps to starting a flood simulation, obtaining the topographic data base and developing the flood hydrology. For the first step, a digital terrain model (DTM) has to be overlaid with a grid system. The Grid Developer System (GDS) processor program will overlay the grid system on a DTM data base and assign elevations to the grid elements. Aerial photography, detailed topographic maps, orthographic photos and digitized mapping can be used to locate important features with respect to the grid system such as streets, buildings, bridges, culverts or other flood conveyance or containment structures. Figure 3 is a flow chart that outlines how the various components interface with each other.

Each flood simulation requires either an inflow flood hydrograph or a rain storm. The discharge inflow points might include the alluvial fan apex or a known discharge location in a river system. FLO-2D can be used to generate the flood hydrograph at a specific location by modeling the rainfall-runoff in the upstream watershed. Another approach is to use an external hydrologic model to generate an inflow hydrograph for the FLO-2D model. Rainfall can also be simulated on the water surface as the flood progresses over the grid system. The model inflow flood volume is the primary factor that determines an area of flood inundation. For that reason, it is suggested that an appropriate effort be spent on the hydrology analysis to support the accuracy of the flood routing simulation.

Results from a FLO-2D flood simulation may include: outflow hydrographs from the grid system; hydrographs and flow hydraulics for each channel element; flood hydrographs and hydraulics for designated floodplain cross sections; maximum flow depths and velocities for all grid elements; changes in bed elevation; and a summary of the inflow, outflow, storage and volume losses in the system. The user can specify the temporal and spatial output detail including the outflow hydrograph locations, the output time intervals and the graphical display of the flood progression over the grid system. Starting with the preliminary FLO-2D runs, the user should test the output options to determine required level of output detail.

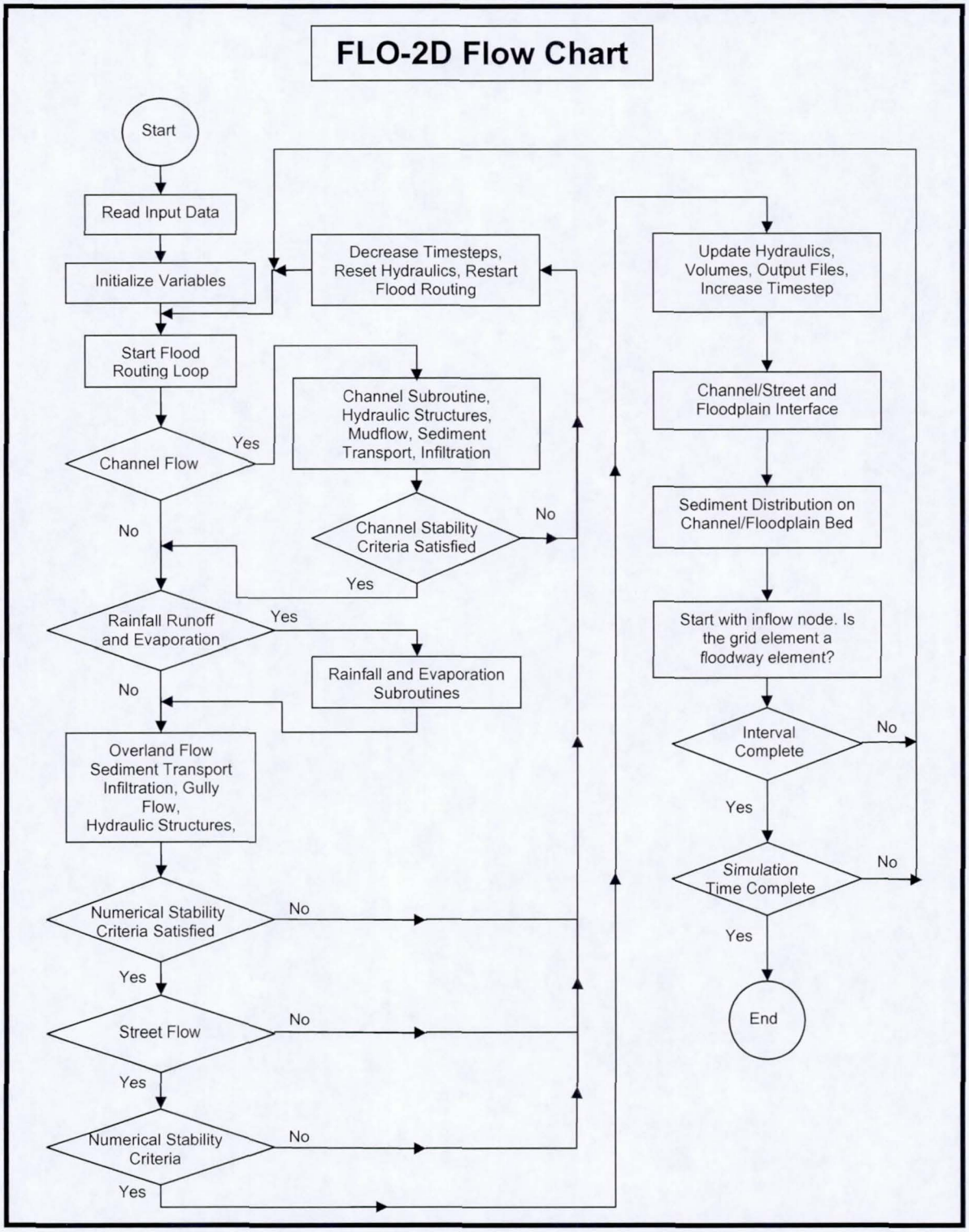


Figure 3. FLO-2D Flow Chart

II. FLO-2D MODEL THEORY

FLO-2D is a simple volume conservation model. It moves the flood volume around on a series of tiles for overland flow or through stream segments for channel routing. Floodwave progression over the flow domain is controlled by topography and resistance to flow. Flood routing in two dimensions is accomplished through a numerical integration of the equations of motion and the conservation of fluid volume for either a water flood or a hyperconcentrated sediment flow. A presentation of the governing equations is followed by a discussion on mud and debris flow modeling.

2.1 Governing Equations

The general constitutive fluid equations include the continuity equation, and the equation of motion (dynamic wave momentum equation):

$$\frac{\partial h}{\partial t} + \frac{\partial h V}{\partial x} = i$$

$$S_f = S_o - \frac{\partial h}{\partial x} - \frac{V}{g}\frac{\partial V}{\partial x} - \frac{V}{g}\frac{\partial V}{\partial x} - \frac{1}{g}\frac{\partial V}{\partial t}$$

where h is the flow depth and V is the depth-averaged velocity in one of the eight flow directions x. The excess rainfall intensity (i) may be nonzero on the flow surface. The friction slope component S_f is based on Manning's equation. The other terms include the bed slope S_o, pressure gradient and convective and local acceleration terms. This equation represents the one-dimensional depth averaged channel flow. For the floodplain, while FLO-2D is multi-direction flow model, the equations of motion in FLO-2D are applied by computing the average flow velocity across a grid element boundary one direction at time. There are eight potential flow directions, the four compass directions (north, east, south and west) and the four diagonal directions (northeast, southeast, southwest and northwest). Each velocity computation is essentially one-dimensional in nature and is solved independently of the other seven directions. The stability of this explicit numerical scheme is based on strict criteria to control the size of the variable computational timestep. The equations representing hyperconcentrated sediment flow are discussed later in the manual.

The relative magnitude of the acceleration components to the bed slope and pressure terms is important. Henderson (1966) computed the relative magnitude of momentum equation terms for a moderately steep alluvial channel and a fast rising hydrograph as follows:

	Bed Slope	Pressure Gradient	Convective Acceleration	Local Acceleration
Momentum Equation Term:	S_o	$\partial h/\partial x$	$V\partial V/g\partial x$	$\partial V/g\partial t$
Magnitude (ft/mi)	26	0.5	0.12 - 0.25	0.05

This illustrates that the application of the kinematic wave ($S_o = S_f$) on moderately steep slopes with relatively steady, uniform flow is sufficient to model floodwave progression and the contribution of the pressure gradient and the acceleration terms can be neglected. The addition of the pressure gradient term to create the diffusive wave equation will enhance overland flow simulation with complex topography. The diffusive wave equation with the pressure gradient is required for floodwave attenuation and change in storage on the floodplain. The local and convective acceleration terms are important to the flood

routing for flat or adverse slopes or very steep slopes or unsteady flow conditions. Only the full dynamic wave equation is applied in FLO-2D model.

2.2 Solution Algorithm - How the Model Works

The differential form of the continuity and momentum equations in the FLO-2D model is solved with a central, finite difference numerical scheme. This explicit algorithm solves the momentum equation for the flow velocity across the grid element boundary one element at a time. The solution of the differential form of the momentum equation results from a discrete representation of the equation when applied at a single point. Explicit schemes are simple to formulate but usually are limited to small timesteps by strict numerical stability criteria. Finite difference schemes can require lengthy computer runs to simulate steep rising or very slow rising floodwaves, channels with highly variable cross sections, abrupt changes in slope, split flow and ponded flow areas.

The solution domain in the FLO-2D model is discretized into uniform, square grid elements. The computational procedure for overland flow involves calculating the discharge across each of the boundaries in the eight potential flow directions (Figure 4) and begins with a linear estimate of the flow depth at the grid element boundary. The estimated boundary flow depth is an average of the flow depths in the two grid elements that will be sharing discharge in one of the eight directions. Non-linear estimates of the boundary depth were attempted in previous versions of the model, but they did not significantly improve the results. Other hydraulic parameters are also averaged between the two grid elements to compute the flow velocity including flow resistance (Manning's n-value), flow area, slope, water surface elevation and wetted perimeter. The flow velocity (dependent variable) across the boundary is computed from the solution of the momentum equation (discussed below). Using the average flow area between two elements, the discharge for each timestep is determined by multiplying the velocity times flow area.

The full dynamic wave equation is a second order, non-linear, partial differential equation. To solve the equation for the flow velocity at a grid element boundary, initially the flow velocity is calculated with the diffusive wave equation using the average water surface slope (bed slope plus pressure head gradient). This velocity is then used as a first estimate (or a seed) in the second order Newton-Raphson tangent method to determine the roots of the full dynamic wave equation (James, et. al., 1977). Manning's equation is applied to compute the friction slope. If the Newton-Raphson solution fails to converge after 3 iterations, the algorithm defaults to the diffusive wave solution.

In the full dynamic wave momentum equation, the local acceleration term is the difference in the velocity for the given flow direction over the previous timestep. The convective acceleration term is evaluated as the difference in the flow velocity across the grid element from the previous timestep. For example, the local acceleration term ($1/g*\partial V/\partial t$) for grid element 251 in the east (2) direction converts to:

$$\Delta(V_t - V_{t-1})_{251}/(g * \Delta t)$$

where V_t is the velocity in the east direction for grid element 251 at time t, V_{t-1} is the velocity at the previous timestep (t-1) in the east direction, Δt is the timestep in seconds, and g is the acceleration due to gravity. A similar construct for the convective acceleration term ($V_x/g*\partial V/\partial x$) can be made where V_2 is the velocity in the east direction and V_4 is the velocity in the west direction for grid element 251:

$$V_2 * \Delta(V_2 - V_4)_{251}/(g * \Delta x)$$

The discharge across the grid element boundary is computed by multiplying the velocity times the cross sectional flow area. After the discharge is computed for all eight directions, the net change in discharge (sum of the discharge in the eight flow directions) in or out of the grid element is multiplied by the timestep to determine the net change in the grid element water volume (see Figure 4). This net change

in volume is then divided by the available surface area (A_{surf} = storage area) on the grid element to obtain the increase or decrease in flow depth Δh for the timestep. The channel routing integration is performed essentially the same way except that the flow depth is a function of the channel cross section geometry and there are usually only one upstream and one downstream channel grid element for sharing discharge.

$$\sum Q_x^{i+1} = Q_n + Q_e + Q_s + Q_w + Q_{ne} + Q_{se} + Q_{sw} + Q_{nw} = A_{surf}\ \Delta h/\Delta t$$

where: Q_x = discharge across one boundary
A_{surf} = surface area of one grid element
$\Delta h/\Delta t$ = change in flow depth in a grid element during one timestep

To summarize, the solution algorithm incorporates the following steps:

1. The average flow geometry, roughness and slope between two grid elements are computed.
2. The flow depth d_x for computing the velocity across a grid boundary for the next timestep (i+1) is estimated from the previous timestep i using a linear estimate (the average depth between two elements).

$$d_x^{i+1} = d_x^i + d_{x+1}^i$$

3. The first estimate of the velocity is computed using the diffusive wave equation. The only unknown variable in the diffusive wave equation is the velocity for overland, channel or street flow.
4. The predicted diffusive wave velocity for the current timestep is used as a seed in the Newton-Raphson solution to solve the full dynamic wave equation for the solution velocity. It should be noted that for hyperconcentrated sediment flows such as mud and debris flows, the velocity calculations include the additional viscous and yield stress terms.
5. The discharge Q across the boundary is computed by multiplying the velocity by the cross sectional flow area. For overland flow, the flow width is adjusted by the width reduction factors (WRFs).
6. The incremental discharge for the timestep across the eight boundaries (or upstream and downstream channel elements) are summed,

$$\Delta Q_x^{i+1} = Q_n + Q_e + Q_s + Q_w + Q_{ne} + Q_{se} + Q_{sw} + Q_{nw}$$

and the change in volume (net discharge x timestep) is distributed over the available storage area within the grid or channel element to determine an incremental increase in the flow depth.
where ΔQ_x is the net change in discharge in the eight floodplain directions for the grid element for the

$$\Delta d_x^{i+1} = \Delta Q_x^{i+1}\ \Delta t\ /A_{surf}$$

timestep Δt between time i and i + 1.

7. The numerical stability criteria is then checked for the new grid element flow depth. If any of the stability criteria are exceeded, the simulation time is reset to the previous simulation time, the timestep increment is reduced, all the previous timestep computations are discarded and the velocity computations begin again.
8. The simulation progresses with increasing timesteps until the stability criteria are exceeded.

In this computation sequence, the grid system inflow discharge and rainfall is computed first, then the channel flow is computed. Next, if streets are being simulated, the street discharge is computed and

finally, overland flow in 8-directions is determined (Figure 5). After all the flow routing for these components has been completed, the numerical stability criteria are tested for every floodplain grid, channel or street element. If stability criteria of any element is exceeded, the timestep is reduced by various functions depending on the previous history of stability success and the computation sequence is restarted. If all the numerical stability criteria are successfully met, the timestep is increased for the next grid system computational sweep. During a sweep of the grid system for a timestep, discharge flux is added to the inflow elements, flow velocity and discharge between grid elements are computed and the change in storage volume in each grid element for both water and sediment are determined. All the inflow volume, outflow volume, change in storage or loss from the grid system area are summed at the end of each time step and the volume conservation is computed. Results are written to the output files or to the screen at user specified output time intervals.

The FLO-2D flood routing scheme proceeds on the basis that the timestep is sufficiently small to insure numerical stability (i.e. no numerical surging). The key to efficient finite difference flood routing is that numerical stability criteria limits the timestep to avoid surging and yet allows large enough timesteps to complete the simulation in a reasonable time. FLO-2D has a variable timestep that varies depending on whether the numerical stability criteria are not exceeded or not. The numerical stability criteria are checked for the every grid element on every timestep to ensure that the solution is stable. If the numerical stability criteria are exceeded, the timestep is decreased and all the previous hydraulic computations for that timestep are discarded. Most explicit schemes are subject to the Courant-Friedrich-Lewy (CFL) condition for numerical stability (Jin and Fread, 1997). The CFL condition relates the floodwave celerity to the model time and spatial increments. The physical interpretation of the CFL condition is that a particle of fluid should not travel more than one spatial increment Δx in one timestep Δt (Fletcher,1990). FLO-2D uses the CFL condition for the floodplain, channel and street routing. The timestep Δt is limited by:

$$\Delta t = C\, \Delta x / (\beta V + c)$$

where:

C is the Courant number ($C \leq 1.0$)
Δx is the square grid element width
V is the computed average cross section velocity
β is a coefficient (5/3 for a wide channel)
c is the computed wave celerity

While the coefficient C can vary from 0.3 to 1.0 depending on the type of explicit routing algorithm, a value of 1.0 is employed in the FLO-2D model to allow the model to have the largest timestep. When C is set to 1.0, artificial or numerical diffusivity is theoretically zero for a linear convective equation (Fletcher, 1990).

For nonlinear equations, it is not possible to completely avoid the artificial diffusivity or numerical dispersion by setting C equal to 1.0 (Fletcher, 1990). For full dynamic wave routing, another set of the numerical stability criteria is applied that was developed by Ponce and Theurer (1982). This criteria is a function of bed slope, specific discharge and grid element size. It is expressed as:

$$\Delta t < \zeta\, S_o\, \Delta x^2 / q_o$$

where q_o is the unit discharge, S_o is the bed slope and ζ is an empirical coefficient. The coefficient ζ was created as a variable unique to the grid element and is adjusted by the model during runtime within a minimum and maximum range set by the user. Similar to the CFS criteria, when this numerical stability is exceeded, the hydraulic computations for that timestep are dumped and the timestep is decreased.

Before the CFL and the full dynamic wave equation numerical stability criteria are evaluated during a FLO-2D simulation, the percent change in depth from the previous timestep for a given grid element is checked. This percent change in depth is used to preclude the need for any additional numerical stability analysis. If the percent change in depth is greater than that specified by the user, the timestep is decreased and all the hydraulic computations for that timestep are voided.

Timesteps generally range from 0.1 second to 30 seconds. The model starts with the a minimum timestep equal to 1 second and increases it until one of the three numerical stability condition is exceeded, then the timestep is decreased. If the stability criteria continue to be exceeded, the timestep is decreased until a minimum timestep is reached. If the minimum timestep is not small enough to conserve volume or maintain numerical stability, then the minimum timestep can be reduced, the numerical stability coefficients can be adjusted or the input data can be modified. The timesteps are a function of the discharge flux for a given grid element and its size. Small grid elements with a steep rising hydrograph and large peak discharge require small timesteps. Accuracy is not compromised if small timesteps are used, but the computational time can be long if the grid system is large.

2.3 The Importance of Volume Conservation

A review of a model flood simulation results begins with volume conservation. Volume conservation is an indication numerical stability and accuracy. The inflow volume, outflow volume, change in storage and infiltration and evaporation losses from the grid system are summed at the end of each time step. The difference between the total inflow volume and the outflow volume plus the storage and losses is a measure of the volume conservation. Volume conservation results are written to the output files or to the screen at user specified output time intervals. Data errors, numerical instability, or poorly integrated components may cause a loss of volume conservation. Any simulation not conserving volume should be revised. It should be noted that volume conservation in any flood simulation is not exact. While some numerical error is introduced by rounding numbers, approximations or interpolations (such as with rating tables), volume should be conserved within a fraction of a percent of the inflow volume. The user must decide on an acceptable level of error in the volume conservation. Most simulations are accurate for volume conservation within a few millionths of a percent. Generally, volume conservation within 0.001 percent or less can be considerate as a successful flood simulation.

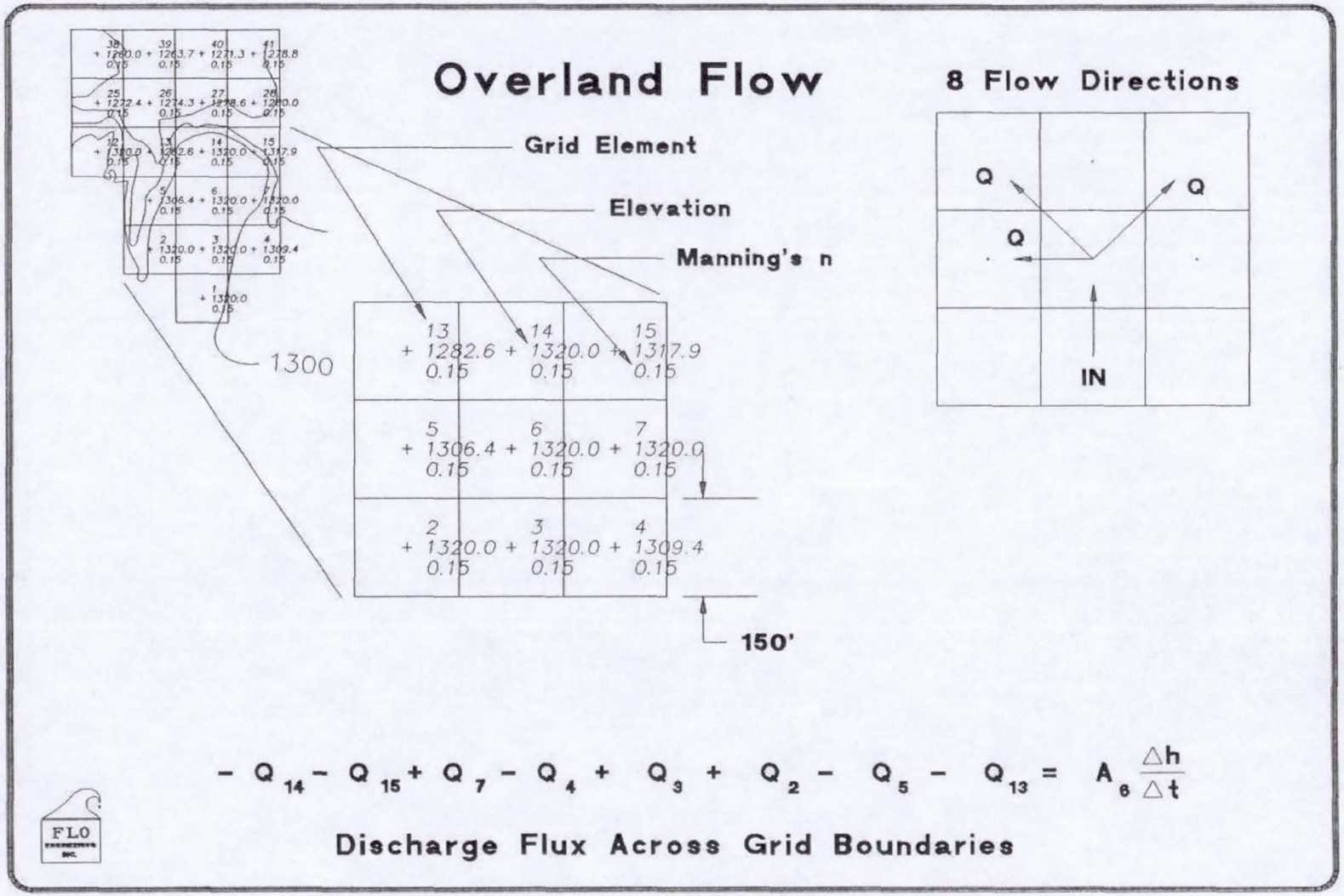


Figure 4. Discharge Flux across Grid Element Boundaries

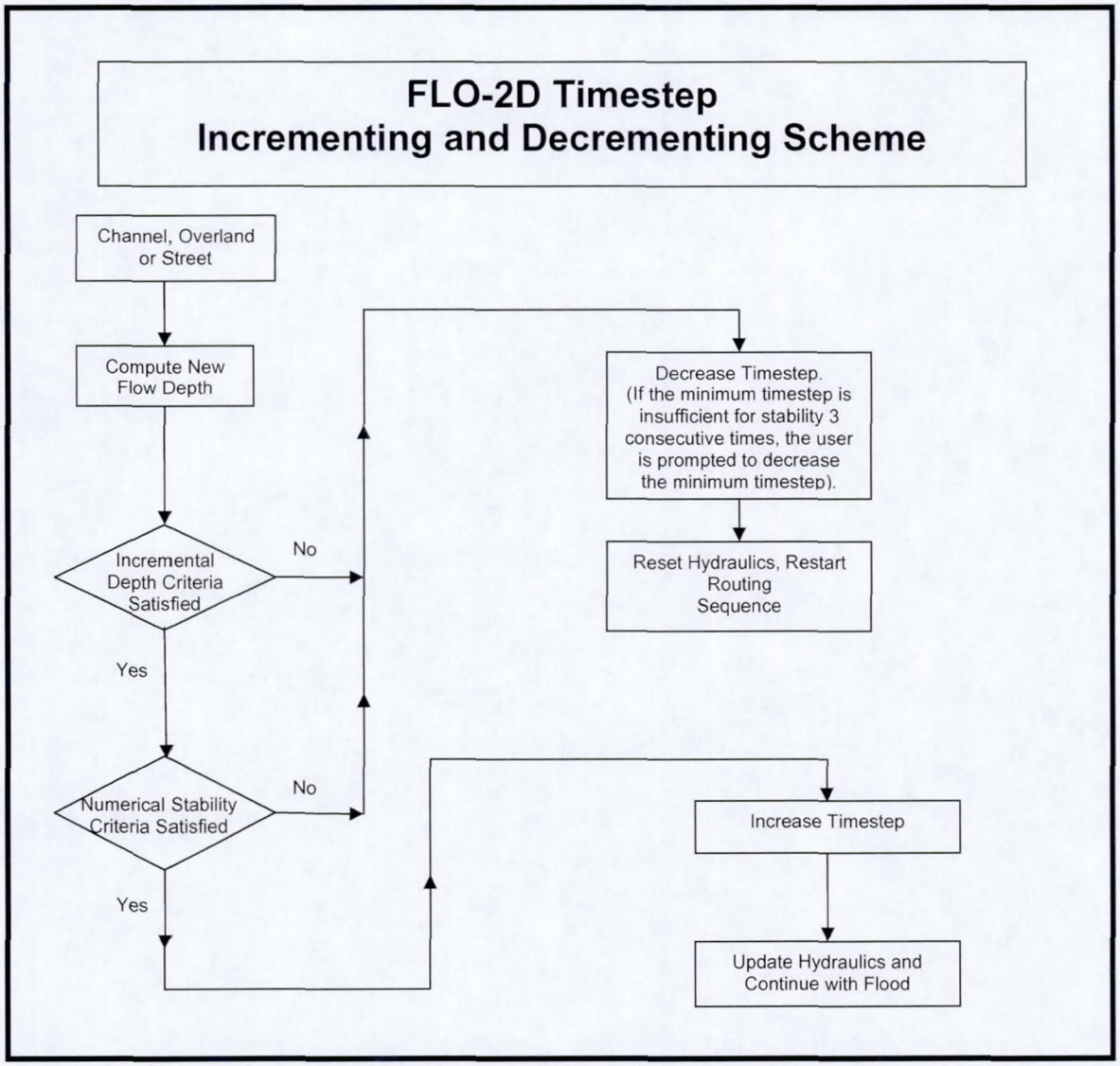


Figure 5. FLO-2D Stability Criteria Flow Chart

III. FLO-2D MODEL SYSTEM

3.1 Assumptions

Conceptualization

FLO-2D flood routing is analyzed using a volume and momentum conservation numerical scheme. The model moves around blocks of fluids on a discretized flow domain consisting of a system of tiles. FLO-2D numerically distributes the volume in finite fluid blocks to mimic the floodwave progression and timing over the discretized surface. Conceptually FLO-2D is not a Lagrangian particle dynamics model but rather a volume conservation model that moves blocks of volume around on the grid system in eight directions while controlled by numerical stability criteria.

Spatial Resolution

The spatial and temporal resolution of the FLO-2D model is dependent on the size of the grid elements and rate of rise in the hydrograph (discharge flux). The rate of change in flood discharge results in an incremental change in the flow depth when distributed over the available grid element surface area for a given timestep. Smaller grid elements may improve the resolution of the flood distribution at the cost of increased computational time, more extensive data files and boundary conditions. A balance must be struck between the number of grid elements and an acceptable computational time. A grid size of 50 ft (15 m) to 500 ft (150 m) is usually appropriate for most simulations. Smaller grid elements will not only significantly increase the number of grid elements (the number of grid elements is quadrupled each time the grid element size is divided by two), but the rate of discharge flux per unit area of the grid element increases.

FLO-2D was developed to simulate large flood events on unconfined surfaces. The discretization of the floodplain topography into a system of square grid elements to accommodate large discharges can obscure some topographic features such as mounds and depressions. This topographic variability will not affect the water surface when the entire valley is flooded. When simulating shallow flow due to steep slopes or small discharge, smaller grid elements should be used. Map resolution and accuracy should be considered when selecting the grid element size. Topographic contour resolution of plus or minus 1 ft (0.3 m) may not support grid elements less than 50 ft (15 m).

For one-dimensional channel flow, the spatial representation and variation in channel geometry is usually limited by the number of cross section surveys. Generally one cross section represents 5 to 10 grid elements. The relationship between flow area, slope and roughness can be distorted by having an insufficient number of cross section surveys. This can result in numerical surges which commonly occur in cases of abrupt channel transitions. The objective is to eliminate any discharge surges without substantially reducing the timestep so that the model runs as fast as possible. This can be accomplished by having gradual transitions between wide and narrow reaches.

Floodwave Attenuation and Discontinuities

Floodwave attenuation in the FLO-2D model occurs in response to flood storage (both channel and overbank). It is the most important feature of the FLO-2D model. Infiltration and evaporation losses can also contribute to floodwave attenuation. Floodwave attenuation represents the interaction of the friction and bed slope terms with the diffusive pressure gradient. While the application of the dynamic wave equation can reduce instabilities in the flood routing computations, rapidly varying flow is still limited by the grid element size. The model does not have the ability to simulate shock waves, rapidly varying flow or hydraulic jumps, and these discontinuities in the flow profile are smoothed out in the model's calculations. Subcritical and supercritical flow transitions are assimilated into the average hydraulic conditions (flow depth and velocity) between two grid elements.

Simulating Ponded Water Conditions

Ponded water conditions may require special consideration. FLO-2D uses Manning's equation to assess hydraulic roughness. Manning's equation is based on uniform, fully developed turbulent flow. In a ponded water condition, the velocity profile may not represent uniform flow. Flow near the bed could be in one-direction and flow near the surface in another direction. A deep ponded water surface might have a very mild or flat slope, but using Manning's equation, high average velocities could still be computed because the velocity is a power function of the depth. It is possible to compute reasonable or accurate water surface elevations in a ponded water condition with FLO-2D, but very small timesteps must be applied. The simplest approach to forcing small timesteps is to set DEPTOL in the TOLER.DAT file to 0.10 or less.

Basic Assumptions

The inherent assumptions in a FLO-2D simulation are:

- Steady flow for the duration of the timestep;
- Hydrostatic pressure distribution;
- Hydraulic roughness is based on steady, uniform turbulent flow resistance;
- A channel element is represented by uniform channel geometry and roughness.

These assumptions are self-explanatory but they remind us that the flow conditions between grid elements are being averaged.

Rigid Bed versus Mobile Bed

When sediment transport is not simulated, a rigid bed is presumed for the flood simulation. Rigid boundary conditions are appropriate for flow over steep slopes, urban flooding and mudflow events. The area of inundation associated with extreme flood events are generally unaffected by bed changes. Channel bed changes generally deviate about a mean condition, and the portion of the flood volume stored in the channel can be small relative the volume on the floodplain. It is assumed in rigid bed simulations that the average flow hydraulics and water surface are not appreciably affected by the scour and deposition that might occur in an individual grid element. Simulating a mobile bed can be more important for smaller floods, for alluvial fan flows and where channel avulsion or sediment deposition might change the flow path.

3.2 Parameter Variability

Roughness Adjustments

For overland flow, there are two flow conditions that warrant special attention. Shallow overland flow where the flow depth is on the order of the roughness elements (>0.2 ft or 0.06 m) can be more effectively modeled by assigning the SHALLOWN parameter in the CONT.DAT file. Suggested n-values for the SHALLOWN parameter range from 0.10 to 0.20. For shallow overland flow less than 0.5 ft (0.15 m) but greater than 0.2 ft (0.06 m), 50% of the SHALLOWN n-value assigned. This roughness adjustment accounts for higher flow resistance associated with shallow flows through vegetation.

Depth variable n-values can be computed for both the channel and floodplain to control the floodwave timing. Roughness n-values to be increased for shallow flows based on the assignment of bankfull n-values for the channel and flows 3 ft (1 m) and higher for overland flooding. The ROUGHADJ variable in the CHAN.DAT file will enable the depth variable n-value adjustment for channel flow. The depth variable n-value is the default condition for floodplain flow and the AMANN variable in the CONT.DAT file will '**turn off**' this adjustment. The basic equation for the roughness n_d as function of flow depth is:

$$n_d = n_b \, r_c \, e^{-(r2 \, depth/dmax)}$$

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = bankfull flow depth
r2 = roughness adjustment coefficient (fixed for overland flow)
$r_c = 1./e^{-r2}$

Ponded Water Conditions

For ponded water conditions with water surface slopes less than 0.001, Manning's open channel flow equation representing the friction slope has limited applicability. In this case, it may necessary to slow the model down by reducing the stability criteria in the TOLER.DAT file. It is recommended that you review the maximum velocities in MAXPLOT for any surging. If you have unreasonable velocities, reduce the stability criteria and/or increase n-values. The selected n-values should be in a range that represents actual flow resistance (see Table 2).

Flow Contraction and Expansion

Flow contraction and expansion between two channel elements is addressed by increasing the head loss as function of the ratio of the flow areas. The head loss coefficient is 0.0 for a ratio of 0.95 or higher. For a contraction of up to 60%, the head loss coefficient varies from 0.0 to 0.6. For flow expansion where the ratio of flows is 60% or less, the head loss coefficient varies from 0.0 to 1.0. The head loss is given by the velocity head $V^2/2g$ times the head loss coefficient and is expressed as slope between the two channel elements. The head loss reduces the available energy gradient between the channel elements. Variability of the contraction and expansion coefficient is automatically computed by the channel routing routine.

Limiting Froude Numbers

Limiting Froude numbers can be specified for overland flow, channel flow and street flow. As an introduction, limiting Froude numbers can be used to adjust the relationship between the flow area, slope and n-values. When the computed Froude number exceeds the limiting Froude number, the n-value is increased for that grid element by a small incremental value for the next timestep. In this manner, the flow can be forced to be subcritical if in reality, critical or supercritical flow is not possible. For example,

in steep-slope sand bed channels, high energy flows may entrain more sediment to sustain subcritical flow. In this case, the limiting Froude number might be set to 0.9. For flow down steep streets, a maximum Froude number of 1.2 to 1.5 may be specified to limit the supercritical flow. Since FLO-2D does not simulate hydraulic jumps, the limiting Froude number should represent average flow conditions in a channel reach. During the falling limb of the hydrograph when the Froude decreases to a value less than 0.5, the flow resistance n-value decreases by a small incremental value until the original n-value is reached. The limiting Froude number will be discussed in more detail in Section 4.6.

Flood Parameter Variability

FLO-2D can simulate the many components of the hydrologic system including rainfall, infiltration, street flow, and flow through hydraulic structures. This level of detail requires a large number of variables. In terms of the channel and floodplain flood routing, the parameters having the greatest effect on the area of inundation or outflow hydrographs are as follows:

- Inflow hydrograph discharge and volume directly affect the area of inundation.
- The overland flow path is primarily a function of the topography.
- The floodplain roughness n-values range from 0.03 to 0.5 and control the overland floodwave speed.
- River channel n-values generally range from 0.020 to 0.085. Roughness adjustment will usually result in only minor variation of the water surface (~ 0.2 ft or 0.06 m).
- The relationship between the channel cross section flow area, bed slope and roughness controls the floodwave routing, attenuation and numerical stability. Flow area has the most important affect on channel routing stability. Changes in the cross section flow area between channel elements should be limited to 25% or less. More cross section surveys may be necessary to simulated rapidly changing flow geometry. Constructed rapid transitions in channel geometry can be modeled, but will require smaller timesteps and more channel detail.
- Floodplain storage loss (ARF values) due to buildings, trees or topography can be globally assigned for the entire grid system using the XARF parameter in the CONT.DAT file. Typically, an XARF value of 5% to 10% can be used to represent a small loss of storage over the entire grid system.
- Most watershed and alluvial fan flooding should be bulked for sediment loading. If the sediment loading will be relatively minor, the XCONC factor in the CONT.DAT file can be used to uniformly bulk all the inflow hydrograph volumes. Typically, watershed flooding that will not generate mudflows can be conservatively bulked using an XCONC value of 10% to 15% by volume. River flood sediment concentration will rarely exceed 5% by volume and setting XCONC = 5% will conservatively bulk the inflow hydrograph volume by 1.05. Mudflow should be simulated by assigning concentrations by volume to the inflow hydrographs and the XCONC factor should not be used.

3.3 Inflow and Outflow Control

A discretized flood hydrograph from an upstream basin can be inflow either to the floodplain, channel or both. More than one grid element can have an inflow hydrograph. Hydrographs can be assigned as either direct inflow or outflow (diversions) from a channel. This could be a simple constant diversion of 100 cfs or a variable hydrograph over the course of the simulation. If mudflows are being simulating then a volumetric sediment concentration or sediment volume must be assigned to each water discharge increment.

For flow out of the grid system, outflow grid elements must be specified for either the floodplain or channel or both. The discharge from outflow elements is equal to sum of the inflows and a flow depth is then assigned to the outflow element based on a weighted average of the upstream flow depths. In this manner, normal flow is approximated at the outflow element. The outflow discharge is totally removed from the system and is accounted to the outflow volume. It is possible to specify outflow from elements that are not on the boundary of the grid system, but outflow elements should be treated as sinks (all the inflow to them is lost from the flow system). Outflow elements should not be modified with ARF's or WRF's, levees, streets, etc. Channel outflow can also be established by a stage-discharge. This option can be used when channel outflow occurs at a hydraulic structure or when a known discharge relationship is available.

Stage-time relationships can be specified for either the floodplain or channel. These relationships can be assigned for outflow elements or for any elements in the system. When a stage-time relationship is specified, volume conservation is accounted for when the discharge enters or leaves the stage-time designed grid element. Stage-time relationships provide opportunity to simulate coastal flooding related to ocean storm surge, hurricane surges or tsunamis (Figure 6). In addition, the backwater effects of tidal variation on river and estuary flooding can be model.

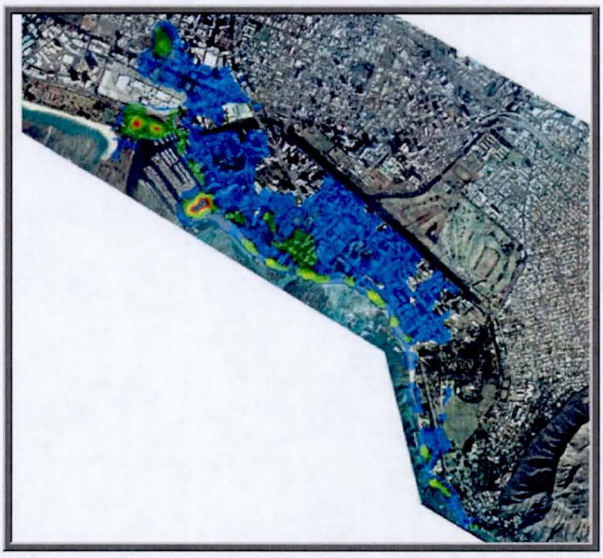


Figure 6. Overland Tsunami Wave Progression in an Urban Area (Waikiki Beach, Hawaii)

3.4 Floodplain Cross Sections

A floodplain cross section analysis can be conducted by specifying grid elements in a cross section in the FPXSEC.DAT file. The grid elements must be contiguous and in a straight line to constitute a cross section across a floodplain or alluvial fan. By designating one or more cross sections, the user can track floodwave attenuation across unconfined surfaces. Both the flood hydrograph and flow

hydraulics can be analyzed at cross sections. The average cross section hydraulics as well as the individual grid element hydraulics in the cross section are summarized in cross section output files.

3.5 Graphical User Interface

A graphical user interface (GUI) facilitates the data input. The GUI creates the ASCII text files used by the FLO-2D model. Specific instructions for the GUI are presented in the Data Input Manual. The GUI is series of forms that represent the individual FLO-2D data files. Each form consists of data dialog boxes, radio switch buttons or grid entry tables. After the data is entered in the GUI dialog boxes, the resulting ASCII text file can be viewed from the GUI or from any other ASCII editor such as MS WordPad®. You can run the model or any of the processor programs from the GUI, but the model doesn't need the GUI to run a simulation.

3.6 Grid Developer System (GDS)

The Grid Developer System (GDS) create and edit the FLO-2D grid system and data files and provides a platform for running the other pre- and post-processor programs. The GDS is a pre-processor program that will overlay the grid system on the DTM points, interpolate and assign elevations to the grid elements. The GDS will then automatically prepare the basic input files for the FLO-2D model. Geo-referenced aerial photos, shape file images or maps can be imported as background images to support the graphical editing.

In addition to developing the FLO-2D grid system, the GDS also provides important editorial features including the assignment of spatially variable grid element attributes such channels, levees, streets, infiltration, area and width reduction factors, floodplain elevation and roughness, inflow and outflow nodes and rill and gully geometry. It allows selection of individual elements or large groups of node using the mouse. Rainfall can also be spatially varied. Detailed instructions are presented in the GDS Manual.

3.7 Graphical Output Options

A graphical display of the flow depths can be viewed on the screen during a FLO-2D simulation to visualize the progression of the floodwave over the potential flow surface. In addition to the predicted flow depths, an inflow hydrograph will be plotted. For rainfall simulation, the cumulative precipitation can also be plotted. The grid element results for floodplain, channel and street flow can be reviewed in a post-processor program MAXPLOT or flood contours can be generated in MAPPER.

Graphical displays are provided in the HYDROG, PROFILES and MAPPER post-processor programs. HYDROG will plot the hydrograph for every channel element. HYDROG can also be used to evaluate the average channel hydraulics in a given reach. The user can select the upstream and downstream channel elements and the program will compute the average of the hydraulics for all the channel elements in the reach including: velocity, depth, discharge, flow area, hydraulic radius, wetted perimeter, top width, width to depth ratio, energy slope, and bed shear stress. The PROFILES program plots channel water surface and bed slopes.

MAPPPER is the primary program for displaying the FLO-2D results. It can create high resolution color contour plots. Several map combinations can be created: grid element or DTM point plots, line contour maps and shaded contour maps. Maps can be created for ground surface elevations, maximum water surface elevations, maximum floodplain flow depths, maximum velocities, maximum static and dynamic pressure, specific energy, and floodway delineation. One of the most important

features of MAPPER is its capability to create flood depth plots using the DTM topographic points. When the user activates the feature, MAPPER will subtract each DTM ground point elevation from the grid element floodplain water surface elevation. The resultant DTM point flow depths can then be interpolated and plotted as color contours. Some of the MAPPER features include:

- Multiple geo-referenced aerial photos in various graphic formats can be imported such as TIFF, BMP, JPG, etc.
- Multiple layer capability including control of layer properties is available.
- Cut and view flow depth and topography profiles.
- Flood damage assessment component to compute the flood damage as function of the FLO-2D predicted maximum depths, building shape files and building value tables (dbf file).
- Flood animation. The floodwave progression over the grid system can be viewed.
- Flood maximum deposition and scour can be plotted.
- Maximum flow velocity vectors can be viewed.
- Hazard maps based on flood intensity and frequency can be created.
- GIS shape files (*.shp) are automatically created with any plotted results. This GIS shape files can be then be imported into ArcView or other GIS programs.
- FEMA Digital Flood Insurance Rate Map (DFIRM) optional tool.

The MAPPER features and functions are described in its own manual.

3.8 Data Output Options

The FLO-2D model has a number of output files to help the user organize the results. Floodplain, channel and street hydraulics are written to file. Hydraulic data include water surface elevation, flow depth and velocities in the eight flow directions. Discharge for specified output intervals (hydrographs) are written to various files. A mass conservation summary table comparing the inflow, outflow and storage in the system is presented in the SUMMARY.OUT file. A complete description of all the output files are presented in the Data Input Manual.

IV. MODEL COMPONENTS

4.1 Model Features

The primary features of the FLO-2D model are:

- Floodwave attenuation can be analyzed with hydrograph routing.
- Overland flow on unconfined surfaces is modeled in eight directions.
- Floodplain flows can be simulated over complex topography and roughness including split flow, shallow flow and flow in multiple channels.
- Channel, street and overland flow and the flow exchange between them can be simulated.
- Channel flow is routed with either a rectangular or trapezoidal geometry or natural cross section data.
- Streets are modeled as shallow rectangular channels.
- The flow regime can vary between subcritical and supercritical.
- Flow over adverse slopes and backwater effects can be simulated.
- Rainfall, infiltration losses and runoff on the alluvial fan or floodplain can be modeled.
- Viscous mudflows can be simulated.
- The effects of flow obstructions such as buildings, walls and levees that limit storage or modify flow paths can be modeled.
- The outflow from bridges and culverts is estimated by user defined rating curves.
- The number of grid and channel elements and most array components is unlimited.

Data file preparation and computer run times vary according to the number and size of the grid elements, the inflow discharge flux and the duration of the inflow flood hydrograph being simulated. Most flood simulations can be accurately performed with square grid elements ranging from 100 ft (30 m) to 500 ft (150 m). Projects have been undertaken with grid elements as small as 10 ft (3 m), although models with grid elements this small tend to be slow. It is important to balance the project detail and the number of model components applied with the mapping resolution and anticipated level of accuracy in the results. It is often more valuable from a project perspective to have a model that runs quickly enabling many simulation scenarios to be performed from which the user can learn about how the project responds to flooding. Model component selection should focus on those physical features that will significantly effect the volume distribution and area of inundation. A brief description of the FLO-2D components follows.

4.2 Overland Flow

The simplest FLO-2D model is overland flow on an alluvial fan or floodplain. The floodplain element attributes can be modified to add detail to the predicted area of inundation. The surface storage area or flow path on grid elements can be adjusted for buildings or other obstructions. Using the area reduction factors (ARFs), a grid element can be completely removed from receiving any inflow. Any of the eight flow directions can be partially or completely blocked to represent flow obstruction. The area of inundation can also be affected by levees, channel breakout flows, flow constriction at bridges and culverts, or street flow in urban areas. Rainfall and infiltration losses can add or subtract from the flow volume on the floodplain surface. These overland flow components are shown in a computational flow chart in Figure 7.

Overland flow velocities and depths vary with topography and the grid element roughness. Spatial variation in floodplain roughness can be assigned through the GDS or FLOENVIR processor. The assignment of overland flow roughness must account for vegetation, surface irregularity, non-uniform and unsteady flow. It is also a function of flow depth. Typical roughness values (Manning's n coefficients) for overland flow are shown in Table 1.

Table 1. Overland Flow Manning's n Roughness Values[1]

Surface	n-value
Dense turf	0.17 - 0.80
Bermuda and dense grass, dense vegetation	0.17 - 0.48
Shrubs and forest litter, pasture	0.30 - 0.40
Average grass cover	0.20 - 0.40
Poor grass cover on rough surface	0.20 - 0.30
Short prairie grass	0.10 - 0.20
Sparse vegetation	0.05 - 0.13
Sparse rangeland with debris	
0% cover	0.09 - 0.34
20 % cover	0.05 - 0.25
Plowed or tilled fields	
Fallow - no residue	0.008 - 0.012
Conventional tillage	0.06 - 0.22
Chisel plow	0.06 - 0.16
Fall disking	0.30 - 0.50
No till - no residue	0.04 - 0.10
No till (20 - 40% residue cover)	0.07 - 0.17
No till (60 - 100% residue cover)	0.17 - 0.47
Open ground with debris	0.10 - 0.20
Shallow glow on asphalt or concrete (0.25" to 1.0")	0.10 - 0.15
Fallow fields	0.08 - 0.12
Open ground, no debris	0.04 - 0.10
Asphalt or concrete	0.02 - 0.05

[1]Adapted from COE, HEC-1 Manual, 1990 and the COE, Technical Engineering and Design Guide, No. 19, 1997 with modifications.

Some FLO-2D projects have been model grid elements inside of the channel. In this case, the channel component is not used and instead the FLO-2D grid system is draped over the channel portion of the topography. While these projects have been conducted with some success, there are several modeling concerns that should be addressed. The FLO-2D model was developed to be able to exchange 1-D channel overbank discharge with the floodplain grid elements. For this reason, the model works well on large flood events and large grid elements. When small grid elements are used inside of a channel with confined flow and large discharges and flow depths, the model may run slow. In addition, there will be zero water surface slope between some grid elements. It should be noted that the application of the Manning's equation for uniform channel to compute the friction slope is no longer valid as the velocity approaches zero (ponded flow condition). The resulting water surface elevations can be accurately predicted but will display some small variation across the channel.

For overland flow, the specific energy, impact pressure and static pressure are computed and reported to file on an output interval basis. The specific energy is computed by adding the flow depth velocity head ($V^2/2g$) to the flow depth. The maximum specific energy is reported to the file SPECENERGY.OUT by grid element. You can use MAPPER to plot the specific energy contours.

The impact pressure P_i for a floodplain grid element is reported as a force per unit length (impact pressure x flow depth). The user can then multiply the impact pressure by the structure length within the grid element to get a maximum impact force on the structure. Impact force is a function of fluid density, structure materials, angle of impact, and a number of other variables. To conservatively estimate the impact pressure, the equation for water taken from Deng (1996):

$$P_i = k\, \rho_f\, V^2$$

where P_i is the impact pressure, coefficient k is 1.28 for both both English and SI units, ρ_f = water density and V is the maximum velocity regardless of direction. For hyperconcentrated sediment flows such as mud floods and mudflows, the fluid density ρ_f and coefficient k is a function of sediment concentration by volume. The coefficient k is based on a regressed relationship as a function of sediment concentration from the data presented in Deng (1996). This relationship is given by,

$$k = 1.261\, e^{Cw}$$

where Cw = sediment concentration by weight. The impact pressure is reported in the file IMPACT.OUT.

The static pressure P_s for each grid element is also expressed as a force per unit length. It is given by the maximum flow depth to the center of gravity $\hat{h}$ times the specific weight of the fluid. The static pressure is then multiplied by the flow depth to compute the static force per unit length of structure (assumes surface area A = l x d). The maximum static pressure is written to the STATICPRESS.OUT file.

$$P_s = \gamma\, \hat{h}$$

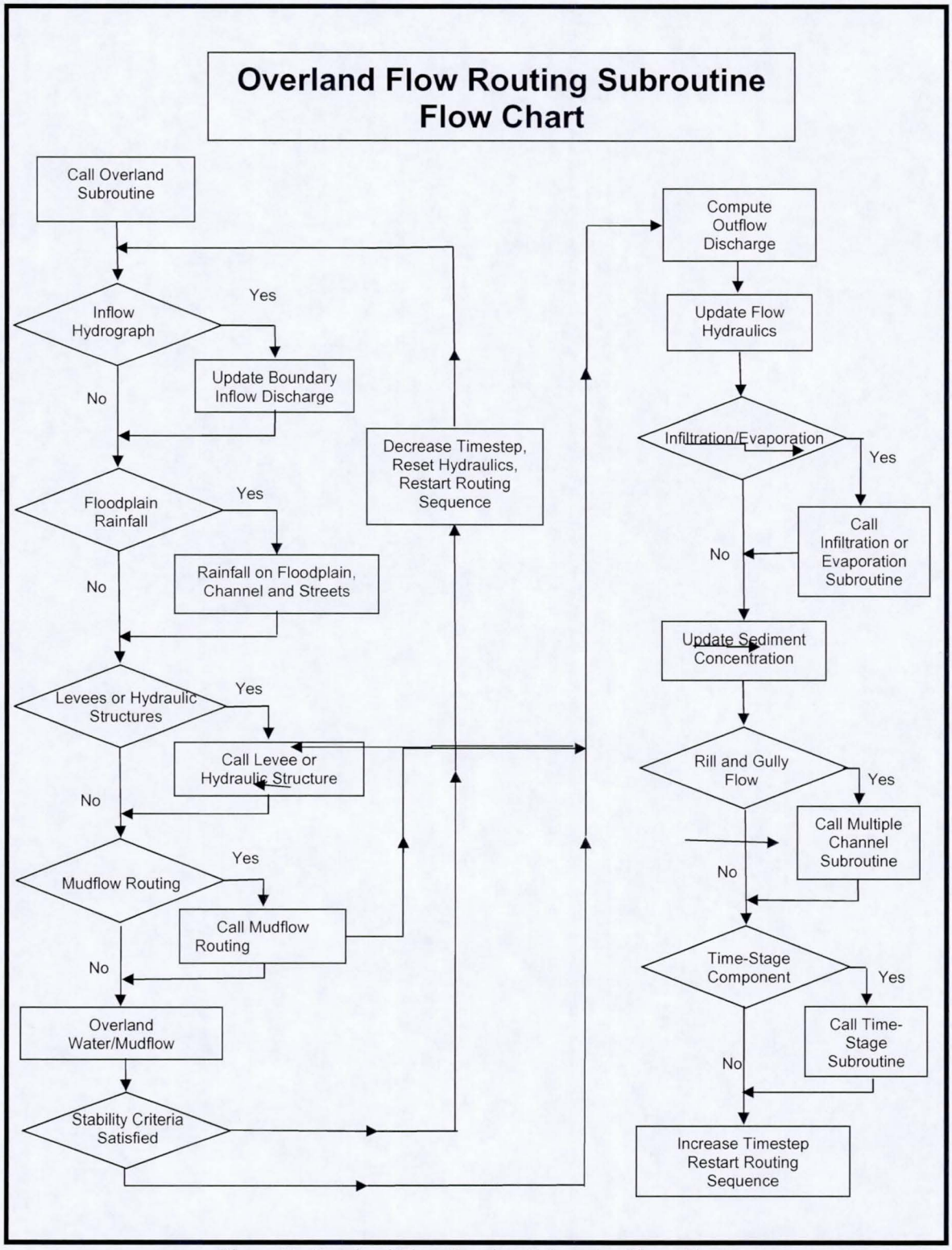


Figure 7. Overland Flow Routing Subroutine Flow Chart

4.3 Channel Flow

Flow in channels is simulated as one-dimensionally. Average flow hydraulics of velocity and depth define the discharge between channel grid elements. Secondary currents, dispersion and superelevation in channel bends are not modeled with the 1-D channel component. The governing equations of continuity and momentum were present in Section 2.1. The average flow path length between two channel elements is on the order of the length of the grid element and this precludes the simulation of hydraulic jumps over a short distance. The flow transition between subcritical and supercritical flow is based on the average conditions between two channel elements.

River channel flow is simulated with either rectangular or trapezoidal or surveyed cross sections and is routed with the dynamic wave approximation to the momentum equation. The channels are represented in the CHAN.DAT by a grid element, cross section geometry that defines the relationship between the thalweg elevation and the bank elevations, average cross section roughness, and the length of channel within the grid element. Channel slope is computed as the difference between the channel element thalweg elevation divided by the half the sum of the channel lengths within the channel elements. Channel elements must be contiguous to be able to share discharge.

The channel width can be larger than the grid element and may encompass several elements (Figure 9). If the channel width is greater than the grid element width, the model extends the channel into neighboring grid elements. A channel may be 1000 ft (300 m) wide and the grid element only 300 ft (100 m) square. The model also makes sure that there is sufficient floodplain surface area after extension. The channel interacts with the right and left bank floodplain elements to share discharge. Each bank can have a unique elevation. If the two bank elevations are different in the CHAN.DAT file, the model automatically splits the channel into two elements even if the channel would fit into one grid element.

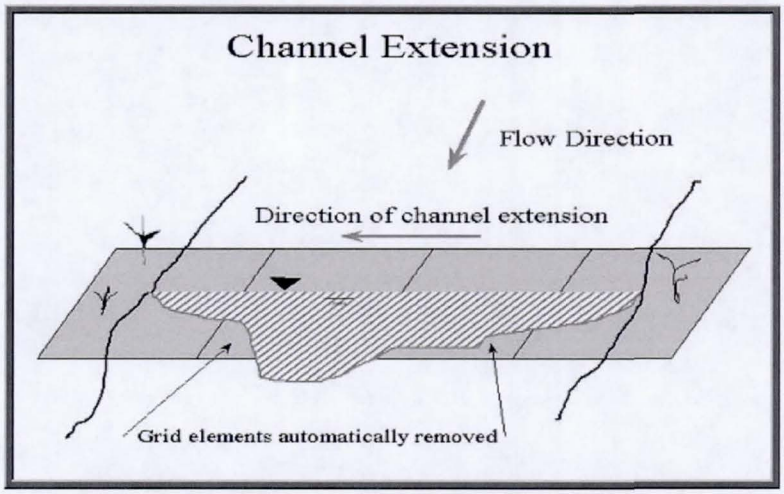


Figure 9. Channel Extension over Several Grid Elements

There are two options for establishing the bank elevation in relationship to the channel bed elevation (thalweg) and the floodplain elevation in the CHAN.DAT file:

1. The channel grid element bed elevation is determined by subtracting the assigned channel thalweg depth from the floodplain elevation.
2. A bank elevation is assigned in the CHAN.DAT file and the channel bed elevation is computed by subtracting the thalweg depth from the lowest bank elevation.

When using cross section data for the channel geometry, option 2 should be applied.

In river simulations, the important components include channel routing, the channel-floodplain interaction, hydraulic structures and levees. These components are described in more detail in the following sections. The basic procedure for creating a FLO-2D river simulation is as follows:

Select Channel Cross Sections. Surveyed river cross sections should be spaced to represent a uniform river reach that may encompass a number of channel elements, say 5 to 10 elements. Geo-referenced surveyed cross section station and elevation data can be entered directly into the model data files or the data can be defined by setting the highest bank to an arbitrary elevation. For channel design purposes, a rectangular or trapezoidal cross section may be selected. To use surveyed cross section data, an XSEC.DAT file has to be created with all cross section station and elevation data. The cross sections are then assigned to a channel element in the CHAN.DAT. The relationship between the flow depth and channel geometry (flow area and wetted perimeter) is based on an interpolation of depth and flow area between vertical slices that constitute a channel geometry rating table for each cross section.

Locate the Channel Element with Respect to the Grid System. Using the GDS and an aerial photo, the channels can be assigned to a grid element. For channel flow to occur through a reach of river, the channel elements must be neighbors.

Adjust the Channel Bed Slope and Interpolate the Cross Sections. Each channel element is assigned a cross section in the CHAN.DAT file. Typically, there are only a few cross sections and many channel elements, so each cross section will be assigned to several channel elements. When the cross sections have all been assigned the channel profile looks like a stair case because the channel elements with the same cross section have identical bed elevations. The channel slope and cross section shape can then be interpolated by using a command in the GDS or in the PROFILES program that adjusts and assigns a cross section with a linear bed slope for each channel element. The interpolated cross section is a weighted flow area adjustment of the cross section to achieve a more uniform rate of change in the flow area.

The user has several other options for setting up the channel data file including grouping the channel elements into segments, specifying initial flow depths, identifying contiguous channel elements that do not share discharge, assigning limiting Froude numbers and depth variable n-value adjustments.

IMPORTANT NOTE: Manning's equation is an empirical formula that was developed on the basis of laboratory and field measurements on steady, uniform, fully developed turbulent flow. Its application, however has become universal for all flow conditions. In a FLO-2D flood simulation the flow is neither steady nor uniform. Channel backwater and ponded flow conditions are two instances when Manning's equation may not be appropriate. The flow resistance should be represented by a composite n-value that includes adjustments to the basic n value for bed irregularities, obstructions, vegetation, variation in channel geometry, channel expansion and contraction, potential rapidly varying flow and variable river planform. Poor selection of n-values or failure to provide spatial variation in roughness can result in numerical surging. Avoid using n-values for natural channels that represent prismatic channel flow.

4.4 Channel-Floodplain Interface

Channel flow is exchanged with the floodplain grid elements in a separate routine after the channel, street and floodplain flow subroutines have been completed. When the channel conveyance capacity is exceeded, an overbank discharge is computed. If the channel flow is less than bankfull discharge and there is no flow on the floodplain, then the channel-floodplain interface routine is not accessed. The channel-floodplain flow exchange is limited by the available exchange volume in the channel or by the available storage volume on the floodplain. The interface routine is internal to the

model and there are no data requirements for its application. This subroutine also computes the flow exchange between the street and the floodplain.

The channel-floodplain exchange is computed for each channel bank element and is based on the potential water surface elevation difference between the channel and the floodplain grid element containing either channel bank (Figure 2). The computed velocity of either the outflow from the channel or the return flow to the channel is computed using the diffusive wave momentum equation. It is assumed that the overbank flow velocity is relatively small and thus the acceleration terms are negligible. The channel bank elevation is established by the surveyed channel geometry and the channel water surface and floodplain water surface is known in relationship to the channel top of bank. For return flow to the channel, if the channel water surface is less than the bank elevation, the bank elevation is used to compute the return flow velocity. Overbank discharge or return flow to the channel is computed using the floodplain assigned roughness. The overland flow can enter a previously dry channel.

4.5 Limiting Froude Numbers

The Froude number represents several physical implications; it delineates subcritical and supercritical flow, it is the ratio of average flow velocity to shallow wave celerity and it relates the movement of a translational wave to the stream flow. Jia (1990) suggested that the trend towards the minimum Froude number is a mechanism that controls the channel adjustment. An alluvial channel system tends to lower its potential energy and attain higher stability as it evolves. This indicates that the greater the bed material movement, the lower the channel stability. It follows therefore that a channel with low bed material movement and high stability will also have minimum hydraulic values. As alluvial channels approach equilibrium conditions, the Froude number will seek a minimum value to reflect minimum bed material motion and maximum channel stability. Since the Froude number identifies a hydraulic state, the most stable condition for sand-bed channel equilibrium may be directly related to a minimum Froude number (Jia, 1990).

Establishing a limiting Froude number in a flood routing model can help sustain the numerical stability. In alluvial channels, the practical range of Froude numbers at bankfull discharge is 0.4 to 0.6. Overland flow on steep alluvial fans can approach critical flow (a Froude number of 1.0). In general, supercritical flow on alluvial fans is suppressed by high rates of sediment transport. High velocities and shallow depths on alluvial surfaces will dissipate energy with sediment entrainment. Supercritical flow is more prevalent on hard surfaces such as bedrock. Jia (1990) provides a relationship to estimate a minimum Froude number (Fr_{min}) for stable alluvial channels at equilibrium:

$$Fr(_{min}) = 4.49\ d^{-0.186}\ (VS)^{0.377}$$

where d = representative sediment size, V = velocity and S = bed slope.

When a limiting Froude is assigned for either floodplain flow (FROUDL in CONT.DAT), street flow (STRFNO in STREET.DAT) or channel flow (FROUDC in CHAN.DAT), the model computes the grid element flow direction Froude number for each timestep. If the limiting Froude number is exceeded, the Manning's n-value for hydraulic flow resistance is increased according to the following criteria.

Percent change from the original n-value	n-value increment increase
< 0.20	0.0002
0.20 < % < 0.50	0.0001
0.50 < % < 1.00	0.00002
1.00 < % < 2.00	0.000002

On the recessional limb, when the limiting Froude number is not exceeded the n-value is decreased by 0.0001. This increase in flow resistance mimics increasing energy loss as the flow accelerates. When the limiting Froude is exceeded, the changes in the n-value are reported in the ROUGH.OUT file. When the simulation is finished the maximum n-values in the ROUGH.OUT file are written to FPLAIN.RGH, CHAN.RGH or STREET.RGH depending on the component utilized. After reviewing the maximum n-value changes in ROUGH.OUT and making any necessary changes in the *.RGH files, these files can be renamed to *.DAT for the next simulation. In this manner, you are spatially calibrating the channel, street and floodplain roughness to a reasonable Froude number.

There is a unique relationship that exists between slope, flow area and roughness. If fact, the Froude number (Fr) is related to the flow resistance K and the energy slope S as given by:

$$Fr = (KS)^{0.5}$$

If there is a mismatch between these physical variables in a flood routing model, then high velocities can occur that may result in flow surging. Assigning a limiting Froude number has several practical advantages. First, it helps to maintain the average flow velocity within a reasonable range. Secondly, a review of the increased n-values in ROUGH.OUT will identify any trouble spots where the velocity exceeds a reasonable value. In this case, the roughness value is increased to offset an inappropriate flow area and slope relationship. When the adjusted n-values in CHAN.RGH and FPLAIN.RGH are used for the next simulation, the effect of the mismatched variables is reduced and numerical surging is dampened. In addition, the increased n-values can prevent oversteepening of the frontal wave. As is the case for any routing model, the best estimate of parameters are not only dependant on the calibration method, but also are governed by the uniqueness and stability of the optimization process. The final n-values used in a simulation should be carefully reviewed for reasonableness. The limiting Froude numbers can be set to "0" for the final simulation to avoid any additional adjustments in the n-values.

4.6 Levees

The FLO-2D levee component confines flow on the floodplain surface by blocking one of the eight flow directions. Levees are designated at the grid element boundaries (Figure 9). If a levee runs through the center of a grid element, the model levee position is represented by one or more of the eight grid element boundaries. Levees often follow the boundaries along a series of consecutive elements. A levee crest elevation can be assigned for each of the eight flow directions in a given grid element. The model will predict levee overtopping. When the flow depth exceeds the levee height, the discharge over the levee is computed using the broadcrested weir flow equation with a 2.85 coefficient. Weir flow occurs until the tailwater depth is 85% of the headwater depth and then at higher flows, the water is exchanged across the levees using the difference in water surface elevation. Levee overtopping will not cause levee failure unless the failure or breach option is invoked.

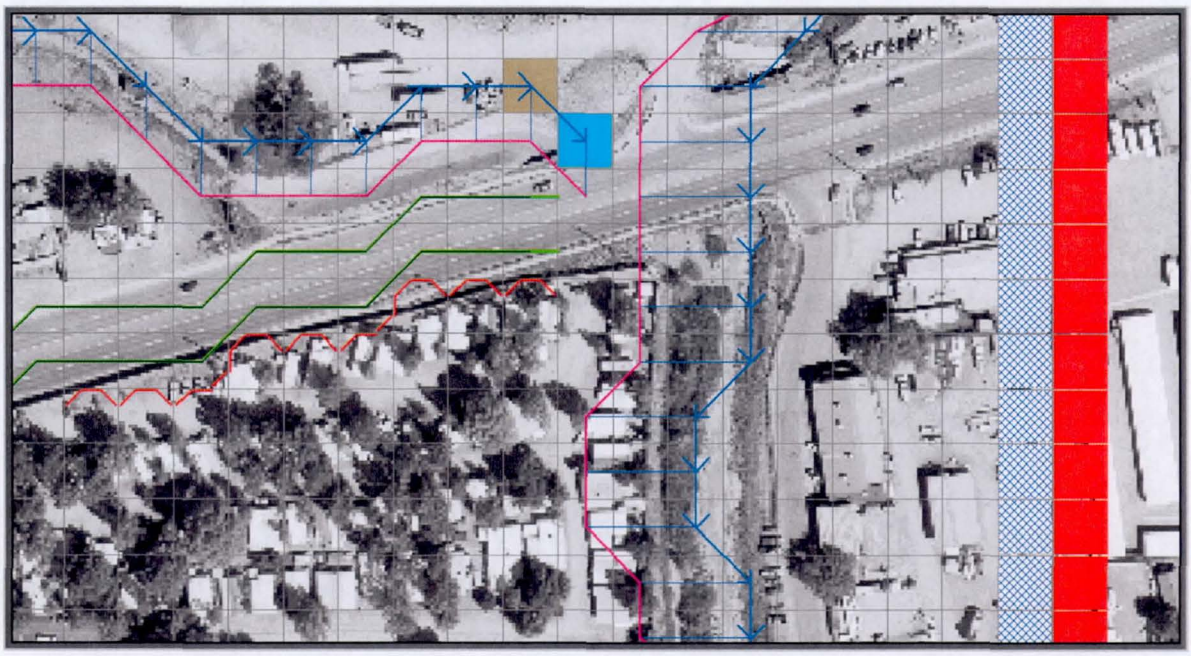


Figure 9. Levees are depicted in Red and the River in Blue in the GDS Program

The levee output files include LEVEE.OUT, LEVOVERTOP.OUT and LEVEEDEFIC.OUT. LEVEE.OUT contains the levee elements that failed. Failure width, failure elevation, discharge from the levee breach and the time of failure occurrence are listed. A discharge hydrograph overtopping the levee element is reported in LEVOVERTOP.OUT. The discharge is combined for all the levee directions that are being overtopped. Finally the LEVEEDIFIC.OUT file lists the levee elements with loss of freeboard during the flood event. Five levels of freeboard deficit are reported:

0 = freeboard > 3 ft (0.9 m)

1 = 2 ft (0.6 m) < freeboard < 3 ft (0.9 m)

2 = 1 ft (0.3 m) < freeboard < 2 ft (0.6 m)

3 = freeboard < 1 ft (0.3 m)

4 = levee is overtopped by flow.

The levee deficit can be displayed graphically in both MAXPLOT and MAPPER.

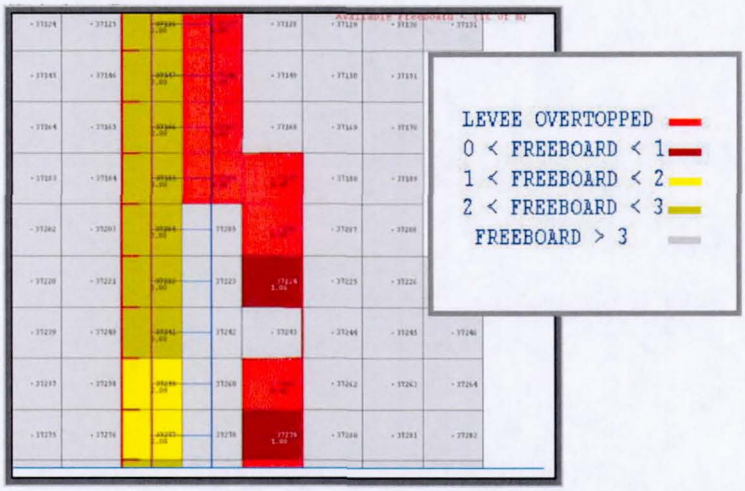


Figure 10. Levee Freeboard Deficit Plot in Mapper

4.7 Levee and Dam Breach Failures

FLO-2D can simulate levee and dam breach failures (Figures 11 and 12). There are two failure modes; one is a simple uniform rate of breach expansion and the other predicts the breach erosion. For both cases, the breach timestep is controlled by the flood routing model. FLO-2D computes the discharge through the breach, the change in upstream storage, the tailwater and backwater effects, and the downstream flood routing. Each failure option generates a series of output files to assist the user in analyzing the response to the dam or levee breach. The model reports of the time of breach or overtopping, the breach hydrograph, peak discharge through the breach, and breach parameters as a function of time. Additional output files that define the breach hazard include the time-to-flow-depth output files that report the time to the maximum flow depth, the time to one foot flow depth and time to two foot flow depth which are useful for delineating evacuation routes.

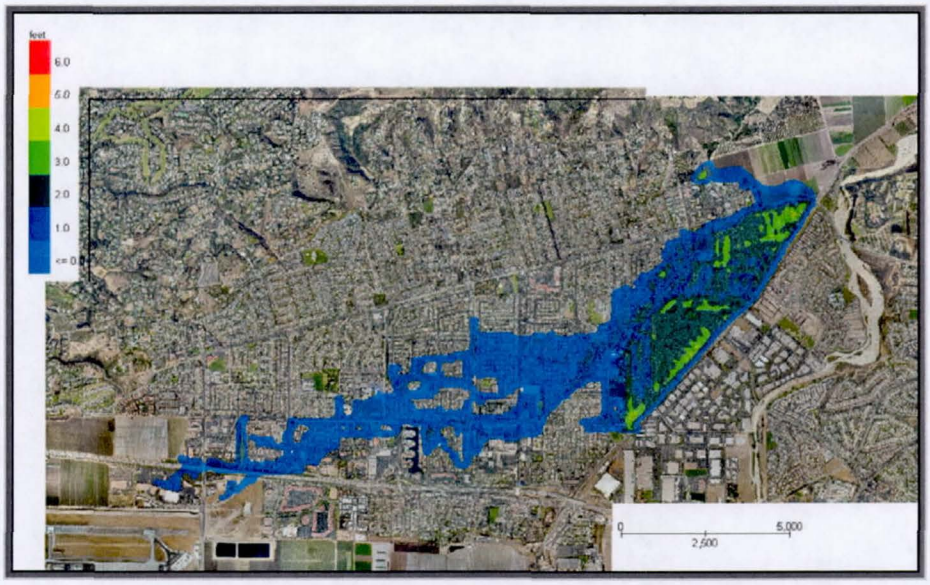


Figure 11. Example of Levee Breach Urban Flooding

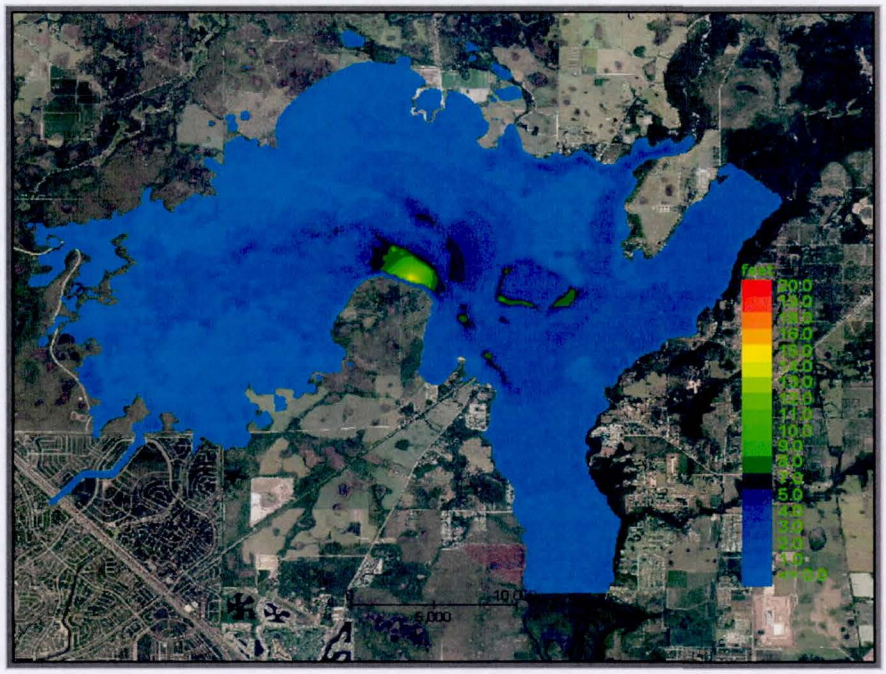


Figure 12. Example of a Proposed Domestic Water Supply Reservoir Breach Failure

For the simplified levee failure method, the breach can enlarge vertically or horizontally. Rates of breach expansion in feet or meters per hour can be specified for both the horizontal and vertical failure modes. A final levee base elevation that is higher than the floodplain elevation can also be specified. The levee failure can occur for the entire grid element width for a given flow direction. Discharge through the breach is based on the breach width and the difference in water surface elevations on the two sides of the levee. Levee failure can also be initiated by a prescribed specified water surface elevation for a given duration. The flow through the breach or overtop the levee is computed as broadcrested weir flow using a weir coefficient of 2.85.

The breach erosion component was added to the FLO-2D model to combine the river-floodplain exchange and unconfined flooding components with a realistic assessment of a levee or dam failure. The BREACH model by Fread (1988) was the basis for the breach component. The BREACH model code was obtained from the National Weather Service and extensively revised. There were a number of code errors in the original Breach model that were corrected.

The basic mechanisms of dam or levee breach failure are overtopping, piping and slope stability failure by sliding, slumping or collapse. In FLO-2D, a dam or levee breach can fail as follows:

- Overtopping and development of a breach channel;
- Piping failure;
- Piping failure and roof collapse and development of a breach channel;
- Breach channel enlargement through side slope slumping;
- Breach enlargement by wedge collapse.

The user has the option to specify the breach element and breach elevation or to assign global parameters and the model will determine which grid element is breached. During a flood simulation, water ponds against the levee or dam until the water surface elevation is higher than the structures and overtops it or exceeds a prescribed elevation for the breach. The global breach elevation can be specified as a depth below the crest elevation. When the water elevation exceeds the breach elevation for a given duration, piping is initiated. If the pipe roof collapse, then the discharge is computed through the resultant breach channel. A breach channel is also simulated if the levee is overtopped. A description of the breach enlargement routine follows.

If the user specifies a breach elevation, then piping will be initiated first when the upstream water surface exceeds the pipe bottom elevation. The breach discharge is computed as weir flow with a user specified weir coefficient. The discharge is then used to compute velocity and depth in a rectangular pipe channel. With the channel hydraulics and dam or levee embankment material parameters, sediment transport capacity is computed using a modified Meyer-Peter Mueller bedload equation (Fread, 1988). The sediment discharge is assumed to erode uniformly from the walls, bed and rood of the pipe (Figure 13). When the pipe height is larger than the material remaining in the embankment of above, the roof of the pipe collapses and breach channel flow ensues. The channel discharge is also calculated by the weir equation and similar to the pipe failure the eroded sediment is distributed on the walls and bed of the rectangular channel (see Figure 14). As the channel width and depth increases, the slope stability is checked and if the stability criteria is exceeded, the sides of the channel slump and the rectangular breach transitions to a trapezoidal channel. The breach continues to wide until the top width of the channel equals or exceeds the octagon width of the grid element. At this point the breach is assumed to be stable.

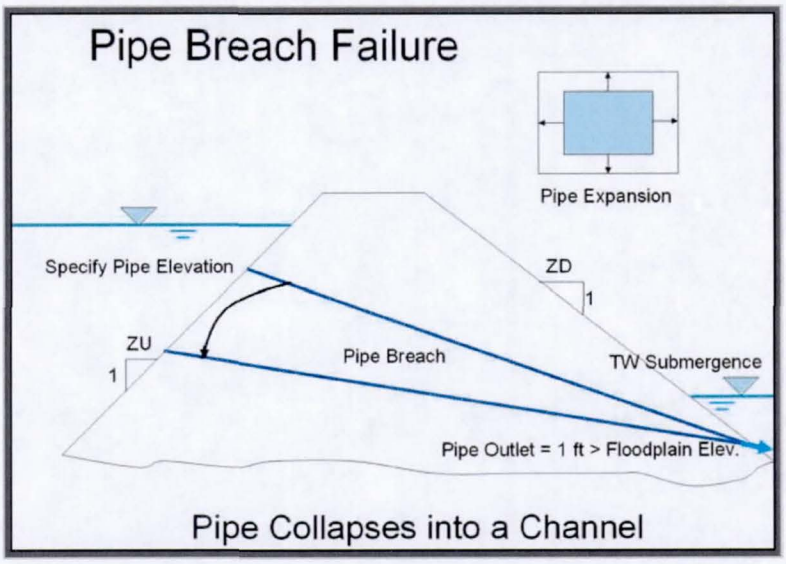


Figure 13. Pipe Breach Failure

Breach enlargement is also possible by sudden collapse of the upper portion of the dam. The collapse would consist of a wedge shaped mass of embankment material. This collapse is caused by the water pressure on the upstream side of the wedge exceeding the friction forces of shear and cohesion that resist sliding. When the breach collapse occurs, it is assumed that the breach enlargement ceases until all the wedge material is transported downstream. A flow chart of the basic computation scheme for the breach component is shown in Figure 15.

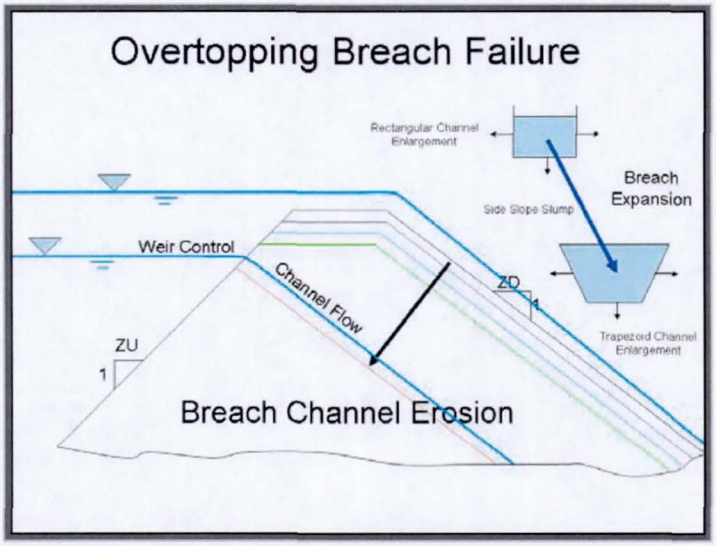


Figure 14. Overtopping and Channel Breach Erosion

Water routed through the breach is accounted by the volume conservation routines in the FLO-2D model that tracks the storage along with the discharge in and out of every grid element according to the FLO-2D timesteps. These timesteps are significantly smaller (5 to 10 times smaller) than the timesteps used in the original BREACH model. The breach component also assesses the sediment volume conservation and the water discharge through the breach is bulked by the sediment eroded during the breach failure. Routing water thru the breach continues until the water surface elevation no longer exceeds the bottom breach elevations or until all the ponded water is gone.

One of the reasons for selecting the BREACH model is that the program had sufficient geotechnical detail to mathematical represent the physical process of dam breach failure. Refer to Fread (1988) for a presentation of the equations and more discussion of the breach theory. The breach model includes the following features:

- The embankment can have an impervious core and a non-cohesive shell with different materials;
- Embankment material properties include sediment size, specific weight, cohesive strength, internal friction angle, porosity and Manning's n-value;
- Breach channel initiation through piping failure;
- Enlargement of the breach through sudden structural collapse or slope instability;
- Riprap material or grass on the downstream face;
- Sediment transport for different size sediment in the embankment core or shell.

There are several important assumptions have been hardwired into the breach model. These are:

- Initial breach width to depth ratio (BRATIO) – if the assigned breach width to depth ration is 0., then BRATIO = 2.
- The initial piping width is assumed to be 0.5 ft (0.15 m).
- The minimum and maximum Manning's n-values permitted for the breach flow resistance are 0.02 and 0.25, respectively.
- The downstream pipe outlet at the toe of the dam or levee is the grid element floodplain elevation plus 1 ft (0.3 m).

- Breach discharge is computed if the upstream water surface elevation exceeds the upstream breach pipe or channel bottom elevation plus the tolerance value (TOL ~ 0.1 ft or 0.3 m).
- If the specified initial breach elevation in the BREACHDATA.DAT file is less than 10.0 ft (3.0 m), then the initial piping breach elevation is assumed to be the dam or levee crest elevation minus the assigned breach elevation (Initial Breach Elevation = Levee Crest – BRBOTTOMEL).

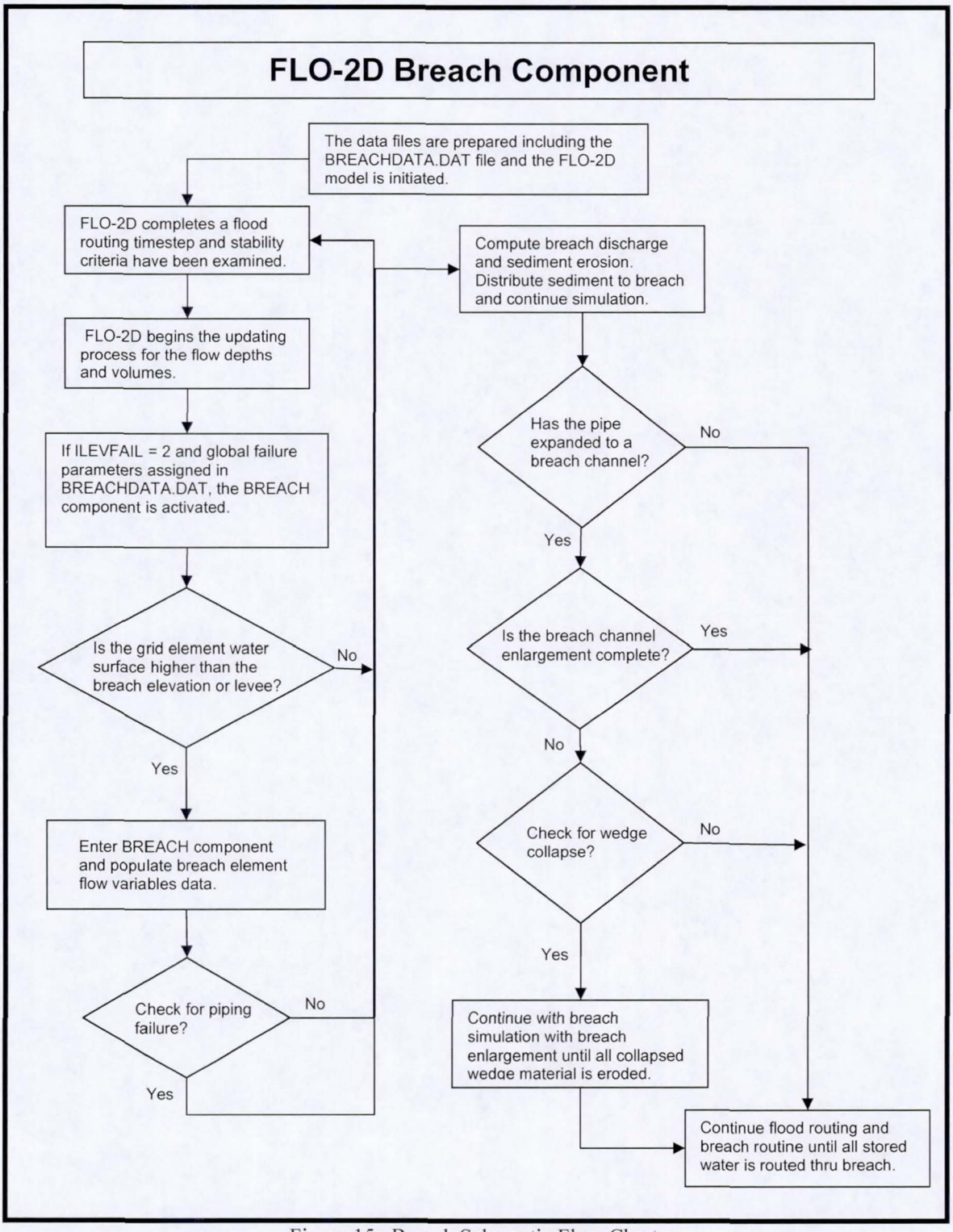


Figure 15. Breach Schematic Flow Chart

4.8 Hydraulic Structures

Hydraulic structures are simulated by specifying either discharge rating curves or rating tables. Hydraulic structures can include bridges, culverts, weirs, spillways or any hydraulic facility that controls conveyance and whose discharge can be specifying by a rating curve or tables. Backwater effects upstream of bridges or culverts as well as blockage of a culvert or overtopping of a bridge can be simulated. A hydraulic structure can control the discharge between channel or floodplain grid elements that do not have to be contiguous but may be separated by several grid elements. For example, a culvert under an interstate highway may span several grid elements.

A hydraulic structure rating curve equation specifies discharge as a function of the headwater depth h:

$$Q = a\,h^b$$

where: (a) is a regression coefficient and (b) is a regression exponent. More than one power regression relationship may be used for a hydraulic structure by specifying the maximum depth for which the relationship is valid. For example, one depth relationship can represent culvert inlet control and a second relationship can be used for the outlet control. In the case of bridge flow, blockage can simulated with a second regression that has a zero coefficient for the height of the bridge low chord.

By specifying a hydraulic structure rating table, the model interpolates between the depth and discharge increments to calculate the discharge. A typical rating curve will start with zero depth and zero discharge and increase in non-uniform increments to the maximum expected discharge. The rating table may be more accurate than the regression relationship if the regression is nonlinear on a log-log plot of the depth and discharge. Flow blockage by debris can be simulated by setting the discharge equal to zero corresponding to a prescribed depth. This blockage option may useful in simulating worst case mud and debris flow scenarios where bridges or culverts are located on alluvial fans. Each bridge on an alluvial fan channel can have simulated blockage forcing all the discharge to flow overland on the fan surface.

4.9 Street Flow

Street flow is simulated as flow in shallow rectangular channels with a curb height using the same routing algorithm as channels. The flow direction, street width and roughness are specified for each street section within the grid element. Street and overland flow exchanges are computed in the channel-floodplain flow exchange subroutine. When the curb height is exceeded, the discharge to floodplain portion of the grid element is computed. Return flow to the streets is also simulated.

Streets are assumed to emanate from the center of the grid element to the element boundary in the eight flow directions (Figure 16). For example, an east-west street across a grid element would be assigned two street sections. Each section has a length of one-half the grid element side or diagonal. A given grid element may contain one or more streets and the streets may intersect. Street roughness values, street widths, elevations and curb heights can be modified on a grid element or street section basis in the GDS program.

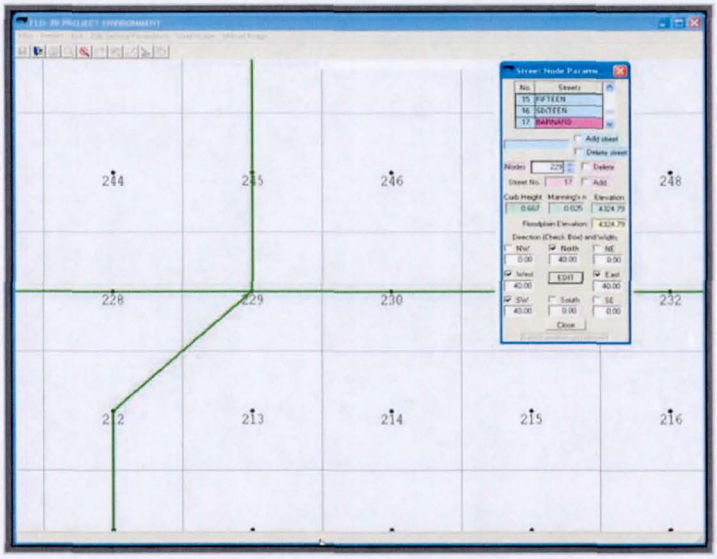


Figure 16. Streets Depicted in Green in the FLOENVIR Program.

4.10 Floodplain Surface Storage Area Modification and Flow Obstruction

One of the unique features the FLO-2D model is its ability to simulate flow problems associated with flow obstructions or loss of flood storage. Area reduction factors (ARFs) and width reduction factors (WRFs) are coefficients that modify the individual grid element surface area storage and flow width. ARFs can be used to reduce the flood volume storage on grid elements due to buildings or topography. WRFs can be assigned to any of the eight flow directions in a grid element and can partially or completely obstruct flow paths in all eight directions simulating floodwalls, buildings or berms.

These factors can greatly enhance the detail of the flood simulation through an urban area. Area reduction factors are specified as a percentage of the total grid element surface area (less than or equal to 1.0). Width reduction factors are specified as a percentage of the grid element side (less than or equal to 1.0). For example, a wall might obstruct 40% of the flow width of a grid element side and a building could cover 75% of the same grid element (Figure 17).

It is usually sufficient to estimate the area or width reduction on a map by visual inspection without measurement. Visualizing the area or width reduction can be facilitated by plotting the grid system over the digitized maps or importing an image in the GDS to locate the buildings and obstructions with respect to the grid system. As a guideline, the area or width reduction factors should be estimated within 10% to 20%. It should be noted that only four width reduction factors need to be specified for the eight possible flow directions. The other four flow directions are assigned automatically by grid element correlation. Two of the specified width reduction factors are for flow across the diagonals. It is possible to specify individual grid elements that are totally blocked from receiving any flow in the ARF.DAT file.

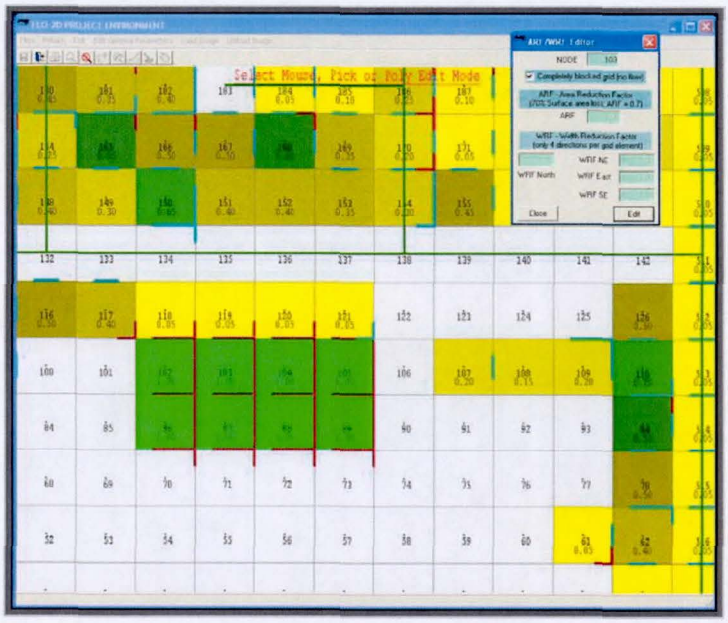


Figure 17. Area and Width Reduction Factors

4.11 Rainfall and Runoff

Rainfall and runoff can be routed to the channel system and then the river flood hydraulics can be computed in the same flood simulation. Either the watershed hydrology or the river hydraulics can be modeled separately with FLO-2D. Alluvial fan or floodplain rainfall can make a substantial contribution to the flood volume and peak discharge. Some fan or floodplain surface areas are similar in size as their upstream watershed areas. In these cases excess rainfall on the fan or floodplain may be equivalent to the volume of inflow hydrograph from the watershed. The rainfall and runoff on the fan or floodplain may precede or lag the arrival of the floodwave from the upstream watershed. It would also dilute the mudflows from the upstream basin.

The storm rainfall is discretized as a cumulative percent of the total. This discretization of the storm hyetograph is established through local rainfall data or through regional drainage criteria that defines storm duration, intensity and distribution. Often in a FLO-2D simulation the first upstream flood inflow hydrograph timestep corresponds to the first rainfall incremental timestep. By altering the storm time distribution on the fan or floodplain, the rainfall can lag or precede the rainfall in the upstream basin depending on the direction of the storm movement over the basin. The storm can also have more or less total rainfall than that occurring in the upstream basin.

There are a number of options to simulate variable rainfall including a moving storm, spatially variable depth area reduction assignment, or even a grid based rain gage data from an actual storm event. Storms can be varied spatially over the grid system with areas of intense or light rainfall. Storms can also move over the grid system by assigning storm speed and direction. A rainfall distribution can be selected from a number of predefined distributions.

Historical storms can be assigned on a grid element basis using real rainfall data. If calibrated Next-Generation Radar (NEXRAD) data is available, the rainfall on the NEXRAD pixels for a given time interval can be automatically interpolated to the FLO-2D grid system using the GDS. Each grid element will be assigned a rainfall total for the NEXRAD time interval and the rainfall is then interpolated by the model for each computational timestep. The result is spatially and temporally variable rainfall-runoff

from the grid system. As example of the application of NEXRAD rainfall on an alluvial fan and watershed near Tucson. Arizona is shown in Figure 18. You can accomplish the same result with gridded network data from a system of rain gages. After the GDS interpolation, each FLO-2D grid element will have a rainfall hyetograph to represent the storm. This is the ultimate temporal and spatial discretization of a storm event and the flood replication has proven to be very accurate.

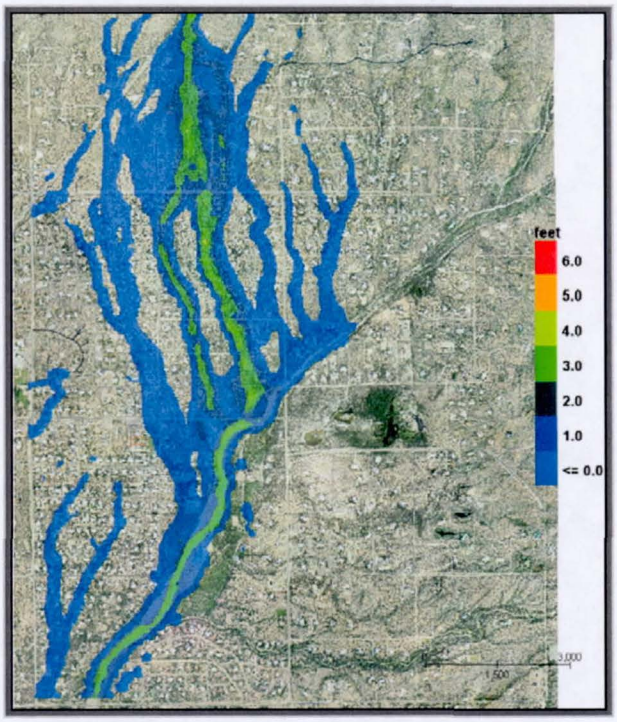


Figure 18. Flooding Replicated from NEXRAD Data near Tucson, Arizona

4.12 Infiltration and Abstraction

Precipitation losses, abstraction (interception) and infiltration are simulated in the FLO-2D model. The initial abstraction is filled prior to simulating infiltration and typical initial abstraction values are presented in Table 2. Infiltration is simulated using either the Green-Ampt infiltration model or the SCS curve number method. The infiltration parameters can be assigned global or the spatial variation of infiltration over the grid system can be modeled by assigning unique hydraulic conductivity and soil suction values to each grid element with the GDS. No infiltration is calculated for assigned streets, buildings or impervious surfaces in the grid elements. Channel infiltration can be also be simulated. Although channel bank seepage is usually only minor portion of the total infiltration losses in the system, it can affect the floodwave progression in an ephemeral channel. The surface area of a natural channel is used to approximate the wetted perimeter to compute the infiltration volume.

The Green-Ampt (1911) equation was selected to compute infiltration losses in the FLO-2D model because it is sensitive to rainfall intensity. When the rainfall exceeds the potential infiltration, then runoff is generated. The infiltration continues after the rainfall has ceased until all the available water has run off or has been infiltrated. The Green-Ampt equation is based on the following assumptions:

- Air displacement from the soil has a negligible effect on the infiltration process;
- Infiltration is a vertical process represented by a distinct piston wetting front;
- Soil compaction due to raindrop impact is insignificant;

- Hysteresis effects of the saturation and desaturation process are neglected;
- Flow depth has limited effect on the infiltration processes.

Table 2. Initial Abstraction

Surface Cover	Abstraction (inches)
Natural[1]	
Desert and rangeland	0.35
Hillslopes Sonoran desert	0.15
Mountain with vegetation	0.25
Developed – Residential[1]	
Lawns	0.20
Desert landscape	0.10
Pavement	0.05
Agricultural fields and pasture	0.50
Conifers[2]	0.01 - 0.36
Hardwoods[2]	0.001 - 0.08
Shrubs[2]	0.01 - 0.08
Grass[2]	0.04 - 0.06
Forest floor[2]	0.02 - 0.44

[1]Maricopa County Drainage Design Manual, 1992.
[2]W. T. Fullerton, Masters Thesis, CSU, 1983

A derivation of the Green-Ampt infiltration modeling procedure can be found in Fullerton (1983). To utilize the Green-Ampt model, hydraulic conductivity, soil suction, volumetric moisture deficiency and the percent impervious area must be specified. Typical hydraulic conductivity, porosity and soil suction parameters are presented in Tables 3 and 4. The volumetric moisture deficiency is evaluated as the difference between the initial and final soil saturation conditions (See Table 5). Depression storage is an initial loss from the potential surface flow (TOL value in TOLER.DAT). This is the amount of water stored in small surface depressions that does not become part of the overland runoff or infiltration.

Table 3. Green Ampt Infiltration - Hydraulic Conductivity and Porosity

Classification	(in/hr)[1]	(in/hr)[2]	(in/hr)[3]	Porosity[4]
sand and loamy sand	1.20	1.21 - 4.14	2.41 - 8.27	0.437
sandy loam	0.40	0.51	1.02	0.437
loam	0.25	0.26	0.52	0.463
silty loam	0.15	0.14	0.27	0.501
silt	0.10			
sandy clay loam	0.06	0.09	0.17	0.398
clay loam	0.04	0.05	0.09	0.464
silty clay loam	0.04	0.03	0.06	0.471
sandy clay	0.02	0.03	0.05	0.430
silty clay	0.02	0.02	0.04	0.479
clay	0.01	0.01	0.02	0.475
very slow			< 0.06[3]	
slow			0.06-.20[3]	
moderately slow			0.20-0.63[3]	
moderate			0.63-2.0[3]	
rapid			2.0-6.3[3]	
very rapid			> 6.3[3]	

[1]Maricopa County Drainage Design Manual, 1992.
[2]James, et. al., Water Resources Bulletin Vol. 28, 1992.
[3]W. T. Fullerton, Masters Thesis, CSU, 1983.
[4]COE Technical Engineering and Design Guide, No. 19, 1997

Table 4. Green Ampt Infiltration - Soil Suction			
Classification	(in)[1]	(in)[2]	(in)[3]
sand and loamy sand	2.4	1.9-2.4	
sandy loam	4.3	4.3	
Loam	3.5	3.5	
silty loam	6.6	6.6	
Silt	7.5		
sandy clay loam	8.6	8.6	
clay loam	8.2	8.2	
silty clay loam	10.8	10.8	
sandy clay	9.4	9.4	
silty clay	11.5	11.5	
Clay	12.4	12.5	
Nickel gravel-sand loam			2.0 - 4.5
Ida silt loam			2.0 - 3.5
Poudre fine sand			2.0 - 4.5
Plainfield sand			3.5 - 5.0
Yolo light clay			5.5 - 10.0
Columbia sandy loam			8.0 - 9.5
Guelph loam			8.0 - 13.0
Muren fine clay			15.0 - 20.0

[1]Maricopa County Drainage Design Manual, 1992.
[2]James, W.P., Warinner, J., Reedy, M., Water Resources Bulletin Vol. 28, 1992.
[3]W. T. Fullerton, Masters Thesis, CSU, 1983.

Table 5. Green Ampt Infiltration -Volumetric Moisture Deficiency		
Classification	Dry (% Diff)	Normal (% Diff)
sand and loamy sand[1]	35	30
sandy loam	35	25
loam	35	25
silty loam	40	25
silt	35	15
sandy clay loam	25	15
clay loam	25	15
silty clay loam	30	15
sandy clay	20	10
silty clay	20	10
clay	15	5

[1]Maricopa County Drainage Design Manual, 1992.

The SCS runoff curve number (CN) loss method is a function of the total rainfall depth and the empirical curve number parameter which ranges from 1 to 100. The rainfall loss is a function of hydrologic soil type, land use and treatment, surface condition and antecedent moisture condition. The method was developed on 24 hour hydrograph data on mild slope eastern rural watersheds in the United

States. Runoff curve numbers have been calibrated or estimated for a wide range of urban areas, agricultural lands and semi-arid range lands. The SCS CN method does not account for variation in rainfall intensity. The method was developed for predicting rainfall runoff from ungaged watersheds and its attractiveness lies in its simplicity. For large basins (especially semi-arid basins) which have unique or variable infiltration characteristics such as channels, the method tends to over-predict runoff (Ponce, 1989).

The SCS curve number parameters can be assigned graphically in the GDS to allow for spatially variable rainfall runoff. Shape files can used to interpolate SCS-CNs from ground cover and soil attributes. The SCS-CN method can be combined with the Green-Ampt infiltration method to compute both rainfall-runoff and overland flow transmission losses. The SCS-CN method will be applied to grid elements with rainfall during the model computational timestep and the Green-Ampt method will compute infiltration for grid elements that do have rainfall during the timestep. This enables transmission losses to be computed with Green-Ampt on alluvial fans and floodplains while the SCS-CN is used to compute the rainfall loss in the watershed basin.

4.13 Evaporation

An open water surface evaporation routine accounts for evaporation losses for long duration floods in large river systems. This component was implemented for the 173 mile Middle Rio Grande model from Cochiti Dam to Elephant Butte Reservoir in New Mexico. The open water surface evaporation computation is based on a total monthly evaporation that is prorated for the number of flood days in the given month. The user must input the total monthly evaporation in inches or mm for each month along with the presumed diurnal hourly percentage of the daily evaporation and the clock time at the start of the flood simulation. The total evaporation is then computed by summing the wetted surface area on both the floodplain and channel grid elements for each timestep. The floodplain wetted surface area excludes the area defined by ARF area reduction factors. The evaporation loss does not include evapotranspiration from floodplain vegetation. The total evaporation loss is reported in the SUMMARY.OUT file and should be compared with the infiltration loss for reasonableness.

4.14 Overland Multiple Channel Flow

The purpose of the multiple channel flow component is to simulate the overland flow in rills and gullies rather than as overland sheet flow. Surface water is often conveyed in small channels, even though they occupy only a fraction of the potential flow area. Simulating rill and gully flow concentrates the discharge and may improve the timing of the runoff routing. The multiple channel routine calculates overland flow as sheet flow within the grid element and flow between the grid elements is computed as rill and gully flow. No overland sheet flow is exchanged between grid elements if both elements have assigned multiple channels. The gully geometry is defined by a maximum depth, width and flow roughness. The multiple channel attributes can be spatially variable on the grid system and can be edited with the GDS program.

If the gully flow exceeds the specified gully depth, the multiple channel can be expanded by a specified incremental width. This channel widening process assumes these gullies are alluvial channels and will widen to accept more flow as the flow reaches bankfull discharge. There is no gully overbank discharge to the overland surface area within the grid element. The gully will continue to widen until the gully width exceeds the width of the grid element, then the flow routing between grid elements will revert to sheet flow. This enables the grid element to be overwhelmed by flood flows. During the falling limb of the hydrograph when the flow depth is less than 1 ft (0.3 m), the gully width will decrease to confine the discharge until the original width is again attained.

4.15 Sediment Transport – Total Load

When a channel rigid bed analysis is performed, any potential cross section changes associated with sediment transport are assumed to have a negligible effect on the predicted water surface. The volume of storage in the channel associated with scour or deposition is relatively small compared to the entire flood volume. This is a reasonable assumption for large river floods on the order of a 100-year flood. For large rivers, the change is flow area associated with scour or deposition will negligible effect on the water surface elevation for flows exceeding the bankfull discharge. On steep alluvial fans, several feet of scour or deposition will usually have a minimal effect on the flow paths of large flood events. For small flood events, the potential effects of channel incision, avulsion, blockage, bank or levee failure and sediment deposition on the flow path should be considered.

To address mobile bed issues, FLO-2D has a sediment transport component that can compute sediment scour or deposition. Within a grid element, sediment transport capacity is computed for either channel, street or overland flow based on the flow hydraulics. The sediment transport capacity is then compared with the sediment supply and the resulting sediment excess or deficit is uniformly distributed over the grid element potential flow surface using the bed porosity based on the dry weight of sediment. For surveyed channel cross sections, a non-uniform sediment distribution relationship is used. There nine sediment transport capacity that can be applied in the FLO-2D model which are discussed below. Each sediment transport formula was derived for unique fluvial geomorphic conditions and the user is encouraged to research the applicability of a selected equation for a particular project. Sediment routing by size fraction and armoring are also options. Sediment continuity is tracked on a grid element basis.

During a FLO-2D flood simulation, the sediment transport capacity is based on the predicted flow hydraulics between floodplain or channel elements, but the sediment transport computation is uncoupled from flow hydraulics. Initially the flow hydraulics are computed for all the grid and channel elements for the given time step and then the sediment transport is computed based on the completed flow hydraulics for that timestep. This assumes that the change in channel geometry resulting from deposition or scour will not have a significant effect on the average flow hydraulics for that timestep. If the scour or deposition is less than 0.10 ft, the sediment storage volume is not distributed on the bed, but is accumulated. Generally it takes several timesteps on the order of 1 to10 seconds to accumulate enough sediment so that the resulting deposition or scour will exceed 0.10 ft (0.03 m). This justifies the uncoupled sediment transport approach.

Sediment routing by size fraction requires a sediment size distribution. A geometric mean sediment diameter is estimated for each sediment interval represented as a percentage of the total sediment sample. Generally, a six or more sediment sizes and the corresponding percentages are determined from a sieve analysis. Each size fraction is routed in the model and the volumes in the bed (floodplain, channel or street) are tracked. The sediment supply to a reach can also be entered in sediment size fractions. An example of sediment data for routing by size fraction is presented below:

Sediment Diameter (mm)	Percent Finer
0.074	0.058
0.149	0.099
0.297	0.156
0.59	0.230
1.19	0.336
2.38	0.492
4.76	0.693
9.53	0.808
19.05	0.913
38.10	1.000

Bed armoring is automatically computed for sediment routing by size fraction. There are no switches to initiate armoring. The armoring process occurs when the upper bed layers of sediment become coarser as the finer sediment is transported out of the bed. An armor layer is complete when the coarse bed material covers the bed and protects the fine sediment below it. To assess armoring, the FLO-2D model tracks the sediment size distribution and volumes in an exchange layer defined by three times the D_{90} grain size of the bed material (Yang, 1996; O'Brien, 1984). The potential armor layer is evaluated each timestep for each element by assessing the volume of each size fraction in the exchange layer (Figure 19).

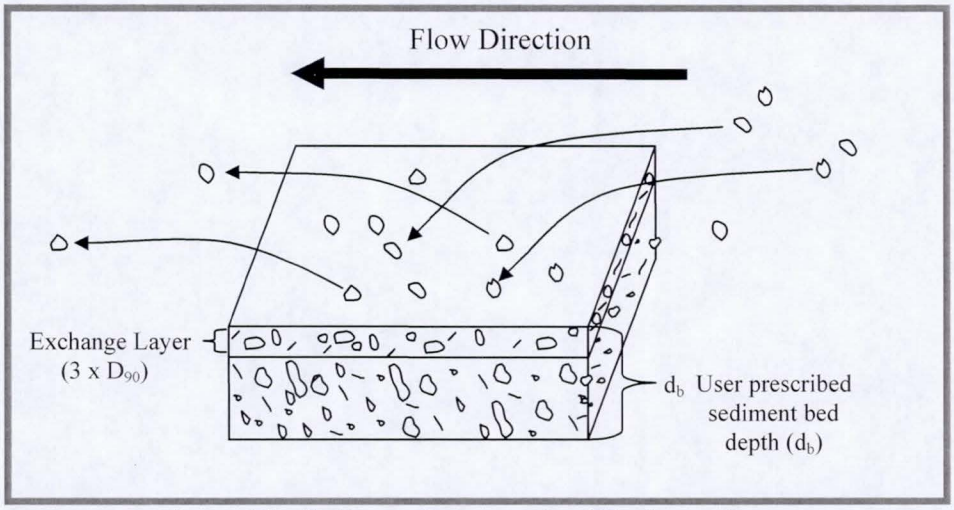


Figure 19. Sediment Transport Bed Exchange Layer

The size fraction percentage is tracked separately in the exchange layer and the rest of the channel bed. When the exchange layer has less than 33% of the original exchange layer volume, the exchange layer is replenished with sediment from the rest of the channel bed using the initial bed material size distribution. This effectively creates an armor layer that is 2 times the D_{90} size of the bed material. As sediment is removed from the exchange layer, the bed coarsens and the size fraction percentage is recomputed. If all smaller sediment size fractions in the exchange layer are removed leaving only the coarse size fraction that the flow cannot transport and the exchange layer thickness is greater than 33% of the original exchange layer thickness, then the bed is armored and no sediment is removed from the bed for that timestep. Sediment deposition can still occur on an armored bed if the supply of a given size fraction to the element exceeds the sediment transport capacity out of the element. The user can specific the total depth of the channel bed available for sediment transport. Sediment scour is limited for adverse slopes to essentially the average reach slope.

FLO-2D calculates the sediment transport capacity using each equation for each grid element and timestep. The user selects only one equation for use in the flood simulation, but can designate one floodplain or channel element to view the sediment transport capacity results for all the equations based on the output interval. The computed sediment transport capacity for each of the nine equations can then be compared by output interval in the SEDTRAN.OUT file. From this file, the range of sediment transport capacity and those equations that appear to be overestimating or underestimated the sediment load can be determined.

Each sediment transport equation is briefly described in the following paragraphs. The user is encouraged to do further research to determine which equation is most appropriate for a specific project channel morphology and hydraulics. When reviewing the SEDTRANS.OUT file, it might be observed that generally

the Ackers-White and Engelund-Hansen equations compute the highest sediment transport capacity; Yang and Zeller-Fullerton result in a moderate sediment transport quantities; and Laursen and Toffaleti calculate the lowest sediment transport capacity. This correlation however varies according to project conditions. A brief discussion of each sediment transport equation in the FLO-2D model follows:

Ackers-White Method. Ackers and White (1973) expressed sediment transport in terms of dimensionless parameters, based on Bagnold's stream power concept. They proposed that only a portion of the bed shear stress is effective in moving coarse sediment. Conversely for fine sediment, the total bed shear stress contributes to the suspended sediment transport. The series of dimensionless parameters are required include a mobility number, representative sediment number and sediment transport function. The various coefficients were determined by best-fit curves of laboratory data involving sediment size greater than 0.04 mm and Froude numbers less than 0.8. The condition for coarse sediment incipient motion agrees well with Sheild's criteria. The Ackers-White approach tends to overestimate the fine sand transport (Julien, 1995) as shown in the above example output.

Engelund-Hansen Method. Bagnold's stream power concept was applied with the similarity principle to derive a sediment transport function. The method involves the energy slope, velocity, bed shear stress, median particle diameter, specific weight of sediment and water, and gravitational acceleration. In accordance with the similarity principle, the method should be applied only to flow over dune bed forms, but Engelund and Hansen (1967) determined that it could be effectively used in both dune bed forms and upper regime sediment transport (plane bed) for particle sizes greater than 0.15 mm.

Karim-Kennedy Equation. The simplified Karim-Kennedy equation (F. Karim, 1998) is used in the FLO-2D model. It is a nonlinear multiple regression equation based on velocity, bed form, sediment size and friction factor using a large number of river flume data sets. The data includes sediment sizes ranging from 0.08 mm to 0.40 mm (river) and 0.18 mm to 29 mm (flume), slope ranging from 0.0008 to 0.0017 (river) and 0.00032 to 0.0243 (flume) and sediment concentrations by volume up to 50,000 ppm. This equation can be used over non-uniform river bed conditions for typical large sand and gravel bed rivers. It will yield results similar to Laursen's and Toffaleti's equations.

Laursen's Transport Function. The Laursen (1958) formula was developed for sediments with a specific gravity of 2.65 and had good agreement with field data from small rivers such as the Niobrara River near Cody, Nebraska. For larger rivers the correlation between measured data and predicted sediment transport was poor (Graf, 1971). This set of equations involved a functional relationship between the flow hydraulics and sediment discharge. The bed shear stress arises from the application of the Manning-Strickler formula. The relationship between shear velocity and sediment particle fall velocity was based on flume data for sediment sizes less than 0.2 mm. The shear velocity and fall velocity ratio expresses the effectiveness of the turbulence in mixing suspended sediments. The critical tractive force in the sediment concentration equation is given by the Shields diagram.

MPM-Smart Equation. This is a modified Meyer-Peter-Mueller (MPM) sediment transport equation (Smart, 1984) for steep channels ranging from 3% to 20%. The original MPM equation underestimated sediment transport capacity because of deficiencies in the roughness values. This equation can be used for sediment sizes greater than 0.4 mm. It was modified to incorporate the effects of nonuniform sediment distributions. It will generate sediment transport rates approaching Englund-Hansen on steep slopes.

MPM-Woo Relationship. For computing the bed material load in steep sloped, sand bed channels such as arroyos, washes and alluvial fans, Mussetter, et al. (1994) linked Woo's relationship for computing the suspended sediment concentration with the Meyer-Peter-Mueller bed-load equation. Woo et al. (1988) developed an equation to account for the variation in fluid properties associated with high sediment concentration. By estimating the bed material transport capacity for a range of hydraulic and bed conditions typical of the Albuquerque, New Mexico area, Mussetter et al. (1994) derived a multiple regression relationship

to compute the bed material load as a function of velocity, depth, slope, sediment size and concentration of fine sediment. The equation requires estimates of exponents and a coefficient and is applicable for velocities up to 20 fps (6 mps), a bed slope < 0.04, a D_{50} < 4.0 mm, and a sediment concentration of less than 60,000 ppm. This equation provides a method for estimating high bed material load in steep, sand bed channels that are beyond the hydraulic conditions for which the other sediment transport equations are applicable.

Toffeleti's Approach. Toffaleti (1969) develop a procedure to calculate the total sediment load by estimating the unmeasured load. Following the Einstein approach, the bed material load is given by the sum of the bedload discharge and the suspended load in three separate zones. Toffaleti computed the bedload concentration from his empirical equation for the lower-zone suspended load discharge and then computed the bedload. The Toffaleti approach requires the average velocity in the water column, hydraulic radius, water temperature, stream width, D_{65} sediment size, energy slope and settling velocity. Simons and Senturk (1976) reported that Toffaleti's total load estimated compared well with 339 river and 282 laboratory data sets.

Yang's Method. Yang (1973) determined that the total sediment concentration was a function of the potential energy dissipation per unit weight of water (stream power) and the stream power was expressed as a function of velocity and slope. In this equation, the total sediment concentration is expressed as a series of dimensionless regression relationships. The equations were based on measured field and flume data with sediment particles ranging from 0.137 mm to 1.71 mm and flows depths from 0.037 ft to 49.9 ft. The majority of the data was limited to medium to coarse sands and flow depths less than 3 ft (Julien, 1995). Yang's equations in the FLO-2D model can be applied to sand and gravel.

Zeller-Fullerton Equation. Zeller-Fullerton is a multiple regression sediment transport equation for a range of channel bed and alluvial floodplain conditions. This empirical equation is a computer generated solution of the Meyer-Peter, Muller bed-load equation combined with Einstein's suspended load to generate a bed material load (Zeller and Fullerton, 1983). For a range of bed material from 0.1 mm to 5.0 mm and a gradation coefficient from 1.0 to 4.0, Julien (1995) reported that this equation should be accurate with 10% of the combined Meyer-Peter Muller and Einstein equations. The Zeller-Fullerton equation assumes that all sediment sizes are available for transport (no armoring). The original Einstein method is assumed to work best when the bedload constitutes a significant portion of the total load (Yang, 1996).

In summary, Yang (1996) made several recommendations for the application of total load sediment transport formulas in the absence of measured data. These recommendations for natural rivers are slightly edited and presented below:

- Use Zeller and Fullerton equation when the bedload is a significant portion of the total load.
- Use Toffaleti's method or the Karim-Kennedy equation for large sand-bed rivers.
- Use Yang's equation for sand and gravel transport in natural rivers.
- Use Ackers-White or Engelund-Hansen equations for subcritical flow in lower sediment transport regime.
- Use Lausen's formula for shallow rivers with silt and fine sand.
- Use MPM-Woo's or MPM-Smart relationships for steep slope, arroyo sand bed channels and alluvial fans.

Yang (1996) reported that ASCE ranked the equations (not including Toffaleti, MPM-Woo, Karin-Kennedy) in 1982 based on 40 field tests and 165 flume measurements in terms of the best overall predictions as follows with Yang ranking the highest: Yang, Laursen, Ackers-White, Engelund-Hansen, and combined Meyer-Peter, Muller and Einstein.

It is important to note that in applying these equations, the wash load is not included in the computations. The wash load should be subtracted from any field data before comparing the field

measurements with the predicted sediment transport results from the equations. It is also important to recognize if the field measurements are sediment supply limited. If this is case, any comparison with the sediment transport capacity equations would be inappropriate.

There are two other sediment transport options available in the FLO-2D model; assignment of rigid bed element and a limitation on the scour depth. Rigid bed element can be used would simulate a concrete apron in a channel below a culvert outlet, channel bed rock or a concrete lined channel reach. The scour depth limitation is a control that can be invoked for sediment routing.

4.16 Mud and Debris Flow Simulation

Very viscous, hyperconcentrated sediment flows are generally referred to as either mud or debris flows. Mudflows are nonhomogeneous, nonNewtonian, transient flood events whose fluid properties change significantly as they flow down steep watershed channels or across alluvial fans. Mudflow behavior is a function of the fluid matrix properties, channel geometry, slope and roughness. The fluid matrix consists of water and fine sediments. At sufficiently high concentrations, the fine sediments alter the properties of the fluid including density, viscosity and yield stress.

There are several important sediment concentration relationships that help to define the nature of hyperconcentrated sediment flows. These relationships relate the sediment concentration by volume, sediment concentration by weight, the sediment density, the mudflow mixture density and the bulking factor. When examining parameters related to mudflows, it is important to identify the sediment concentration as a measure of weight or volume. The sediment concentration by volume C_v is given by:

$$C_v = \text{volume of the sediment/(volume of water plus sediment)}$$

C_v is related to the sediment concentration by weight C_w by:

$$C_v = C_w\gamma / \{\gamma_s - C_w(\gamma_s - \gamma)\}$$

where γ = specific weight of the water and γ_s = specific weight of the sediment. The sediment concentration can also be expressed in parts per million (ppm) by dividing the concentration by weight C_w by 10^6. The specific weight of the mudflow mixture γ_m is a function of the sediment concentration by volume:

$$\gamma_m = \gamma + C_v(\gamma_s - \gamma)$$

Similarly the density of the mudflow mixture ρ_m is given by:

$$\rho_m = \rho + C_v(\rho_s - \rho)$$

and

$$\rho_m = \gamma_m / g$$

where g is gravitational acceleration. Finally, the volume of the total mixture of water and sediment in a mudflow can be determined by multiplying the water volume by the bulking factor. The bulking factor is simply:

$$BF = 1/(1 - C_v)$$

The bulking factor is 2.0 for a sediment concentration by volume of 50%. A sediment concentration of 7% by volume for a conventional river bedload and suspended results in a bulking factor of 1.075 indicating that the flood volume is 7.5% greater than if the flood was considered to be only water.

These basic relationships will be valuable when analyzing mudflow simulations. Most mudflow studies require estimates of the sediment concentration by volume and the bulking factor to describe the magnitude of the event. Average and peak sediment concentrations for the flood hydrograph are important variables for mitigation design.

The full range of sediment flows span from water flooding to mud floods, mudflows and landslides. The distinction between these flood events depends on sediment concentration measured either by weight or volume (Figure 20). Sediment concentration by volume expressed as a percentage is the most commonly used measure. Table 6 lists the four different categories of hyperconcentrated sediment flows and their dominant flow characteristics. This table was developed from the laboratory data using actual mudflow deposits. Some variation in the delineation of the different flow classifications should be expected on the basis of the geology.

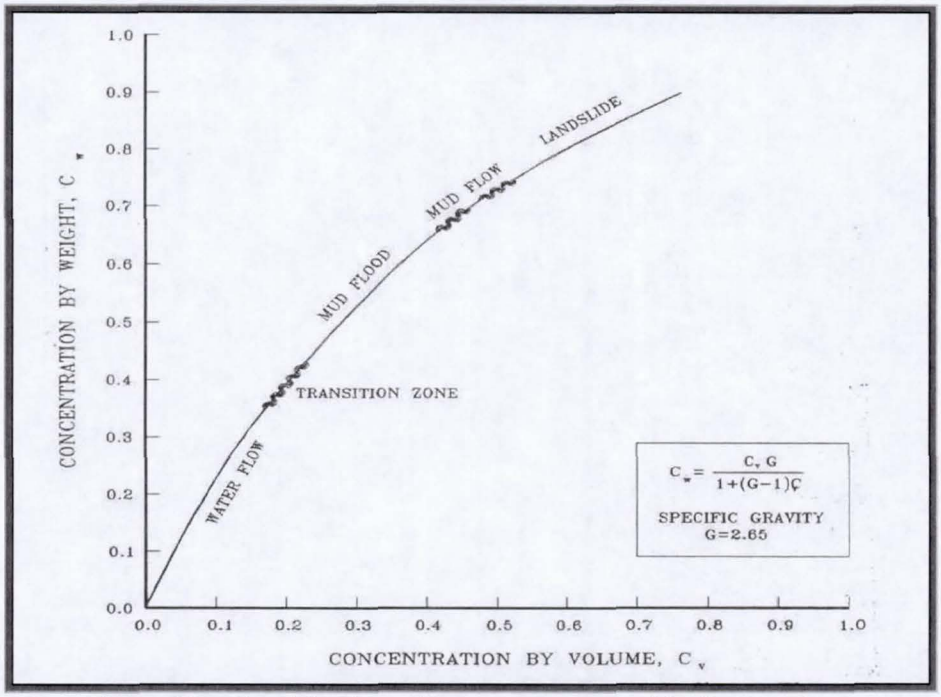


Figure 20. Classification of Hyperconcentrated Sediment Flows

Initial attempts to simulate debris flows were accomplished with one-dimensional flow routing models. DeLeon and Jeppson (1982) modeled laminar water flows with enhanced friction factors. Spatially varied, steady-state Newtonian flow was assumed and flow cessation could not be simulated. Schamber and MacArthur (1985) created a one-dimensional finite element model for mudflows using the Bingham rheological model to evaluate the shear stresses of a nonNewtonian fluid. O'Brien (1986) designed a one-dimensional mudflow model for watershed channels that also utilized the Bingham model. In 1986, MacArthur and Schamber presented a two-dimensional finite element model for application to simplified overland topography (Corps, 1988). The fluid properties were modeled as a Bingham fluid whose shear stress is a function of the fluid viscosity and yield strength.

Takahashi and Tsujimoto (1985) proposed a two-dimensional finite difference model for debris flows based a dilatant fluid model coupled with Coulomb flow resistance. The dilatant fluid model was derived from Bagnold's dispersive stress theory (1954) that describes the stress resulting from the collision of sediment particles. Later, Takahashi and Nakagawa (1989) modified the debris flow model to include turbulence.

Table 6. Mudflow Behavior as a Function of Sediment Concentration			
	Sediment Concentration		Flow Characteristics
	by Volume	by Weight	
Landslide	0.65 - 0.80	0.83 - 0.91	Will not flow; failure by block sliding
	0.55 - 0.65	0.76 - 0.83	Block sliding failure with internal deformation during the slide; slow creep prior to failure
Mudflow	0.48 - 0.55	0.72 - 0.76	Flow evident; slow creep sustained mudflow; plastic deformation under its own weight; cohesive; will not spread on level surface
	0.45 - 0.48	0.69 - 0.72	Flow spreading on level surface; cohesive flow; some mixing
Mud Flood	0.40 - 0.45	0.65 - 0.69	Flow mixes easily; shows fluid properties in deformation; spreads on horizontal surface but maintains an inclined fluid surface; large particle (boulder) setting; waves appear but dissipate rapidly
	0.35 - 0.40	0.59 - 0.65	Marked settling of gravels and cobbles; spreading nearly complete on horizontal surface; liquid surface with two fluid phases appears; waves travel on surface
	0.30 - 0.35	0.54 - 0.59	Separation of water on surface; waves travel easily; most sand and gravel has settled out and moves as bedload
	0.20 - 0.30	0.41 - 0.54	Distinct wave action; fluid surface; all particles resting on bed in quiescent fluid condition
Water Flood	< 0.20	< 0.41	Water flood with conventional suspended load and bedload

O'Brien and Julien (1988), Julien and Lan (1991), and Major and Pierson (1992) investigated mudflows with high concentrations of fine sediment in the fluid matrix. These studies showed that mudflows behave as Bingham fluids with low shear rates ($<10\ s^{-1}$). In fluid matricies with low sediment concentrations, turbulent stresses dominate in the core flow. High concentrations of non-cohensive particles combined with low concentrations of fine particles are required to generate dispersive stress. The quadratic shear stress model proposed by O'Brien and Julien (1985) describes the continuum of flow regimes from viscous to turbulent/dispersive flow.

Hyperconcentrated sediment flows involve the complex interaction of fluid and sediment processes including turbulence, viscous shear, fluid-sediment particle momentum exchange, and sediment particle collision. Sediment particles can collide, grind, and rotate in their movement past each other. Fine sediment cohesion controls the nonNewtonian behavior of the fluid matrix. This cohesion contributes to the yield stress τ_y which must be exceeded by an applied stress in order to initiate fluid motion. By combining the yield stress and viscous stress components, the well-known Bingham rheological model is prescribed.

For large rates of shear such as might occur on steep alluvial fans (10 s^{-1} to 50 s^{-1}), turbulent stresses may be generated. In turbulent flow an additional shear stress component, the dispersive stress, can arise from the collision of sediment particles. Dispersive stress occurs when non-cohesive sediment particles dominate the flow and the percentage of cohesive fine sediment (silts and clays) is small. With increasing high concentrations of fine sediment, fluid turbulence and particle impact will be suppressed and the flow will approach being laminar. Sediment concentration in a given flood event can vary dramatically and as a result viscous and turbulent stresses may alternately dominate, producing flow surges.

FLO-2D routes mudflows as a fluid continuum by predicting viscous fluid motion as function of sediment concentration. A quadratic rheologic model for predicting viscous and yield stresses as function of sediment concentration is employed and sediment continuity is observed. As sediment concentration changes for a given grid element, dilution effects, mudflow cessation and the remobilization of deposits are simulated. Mudflows are dominated by viscous and dispersive stresses and constitute a very different phenomenon than those processes of suspended sediment load and bedload in conventional sediment transport. In should be noted that the sediment transport and mudflow components **cannot** be used together in a FLO-2D simulation.

The shear stress in hyperconcentrated sediment flows, including those described as debris flows, mudflows and mud floods, can be calculated from the summation of five shear stress components. The total shear stress τ depends on the cohesive yield stress τ_c, the Mohr-Coulomb shear τ_{mc}, the viscous shear stress τ_v (η dv/dy), the turbulent shear stress τ_t, and the dispersive shear stress τ_d.

$$\tau = \tau_c + \tau_{mc} + \tau_v + \tau_t + \tau_d$$

When written in terms of shear rates (dv/dy), the following quadratic rheological model can be defined (O'Brien and Julien, 1985):

$$\tau = \tau_y + \eta \left(\frac{dv}{dy}\right) + C \left(\frac{dv}{dy}\right)^2$$

where

$$\tau_y = \tau_c + \tau_{mc}$$

and

$$C = \rho_m \, l^2 + f \, (\rho_m, C_v) \, d_s^2$$

In these equations η is the dynamic viscosity; τ_c is the cohesive yield strength; the Mohr Coulomb stress $\tau_{mc} = p_s \tan\varphi$ depends on the intergranular pressure p_s and the angle of repose φ of the material; C denotes the inertial shear stress coefficient, which depends on the mass density of the mixture ρ_m, the Prandtl mixing length l, the sediment size d_s and a function of the volumetric sediment concentration C_v. Bagnold (1954) defined the function relationship $f(\rho_m, C_v)$ as:

$$f(\rho_m, C_v) = a_i \, \rho_m \left[\left(\frac{C_*}{C_v}\right)^{1/3} - 1 \right]$$

where a_i (~ 0.01) is an empirical coefficient and C_* is the maximum static volume concentration for the sediment particles. It should be noted that Takahashi (1979) found that the coefficient a_i may vary over several orders of magnitude. Egashira et al. (1989) revised this relationship and suggested the following:

$$f(\rho_s, C_v) = \frac{\pi}{12}\left(\frac{6}{\pi}\right)^{1/3} \sin^2 \alpha_I \ \rho_s (1 - e_n^2)\ C_v^{1/3}$$

where the energy restitution coefficient e_n after impact ranges $0.70 < e_n < 0.85$ for sands, α_I is the average particle impact angle and ρ_s is the mass density of sediment particles.

The first two stress terms in the quadratic rheological model are referred to as the Bingham shear stresses (Figure 21). The sum of the yield stress and viscous stress define the total shear stress of a cohesive, mudflow in a viscous flow regime. The last term is the sum of the dispersive and turbulent shear stresses and defines an inertial flow regime for a mud flood. This term is a function of the square of the velocity gradient. A discussion of these stresses and their role in hyperconcentrated sediment flows can be found in Julien and O'Brien (1987, 1993).

A mudflow model that incorporates only the Bingham stresses and ignores the inertial stresses assumes that the simulated flow is dominated by viscous stresses. This assumption is not universally appropriate because all mud floods and some mudflows are very turbulent with velocities as high as 25 fps (8 m/s). Even mudflows with concentrations up to 40% by volume can be turbulent (O'Brien, 1986). Depending on the fluid matrix properties, the viscosity and yield stresses for high sediment concentrations can still be relatively small compared to the turbulent stresses. If the flow is controlled primarily by the viscous stress, it will result in lower velocities. Conversely, if the viscosity and yield stresses are small, the turbulent stress will dominate and the velocities will be higher.

To delineate the role of turbulent and dispersive forces in sand and water mixtures, Hashimoto (1997) developed simplified criteria involving only flow depth d and sediment size D_i. When $d/D_i < 30$, the intergranular forces are dominant. If $d/D_i > 100$, inertial forces dominate. In the range $30 < d/D_i < 100$ both forces play an important role in the momentum exchange. It should be noted, however, that sediment concentration is a critical factor that is not accounted for in this criteria.

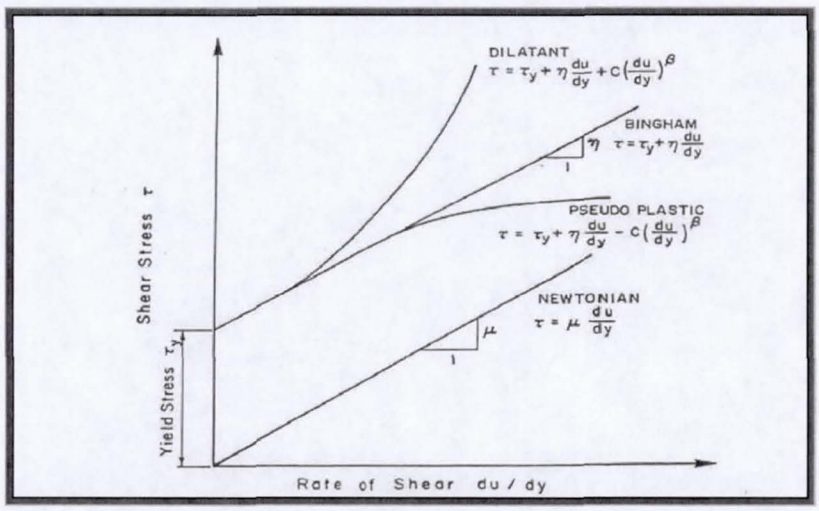


Figure 21. Shear Stress as a Function of Shear Rate for Fluid Deformation Models

To define all the shear stress terms for use in the FLO-2D model, the following approach was taken. By analogy, from the work of Meyer-Peter and Müller (1948) and Einstein (1950), the shear stress relationship is depth integrated and rewritten in the following form as a dimensionless slope:

$$S_f = S_y + S_v + S_{td}$$

where the total friction slope S_f is the sum of the yield slope S_y, the viscous slope S_v, and the turbulent-dispersive slope S_{td}. The viscous and turbulent-dispersive slope terms are written in terms of depth-averaged velocity V. The viscous slope can be written as:

$$S_v = \frac{K\ \eta}{8\ \gamma_m}\ \frac{V}{h^2}$$

where γ_m is the specific weight of the sediment mixture. The resistance parameter K for laminar flow equals 24 for smooth wide rectangular channels but increases significantly (~ 50,000) with roughness and irregular cross section geometry. In Table 7 for Kentucky Blue Grass with a slope of 0.01, K was estimated at 10,000 (Chen, 1976). A value of K = 2,285 was calibrated on the Rudd Creek, Utah mudflow for a residential area and has been used effectively for most urban studies. For laminar and transitional flows, turbulence is suppressed and the laminar flow resistance parameter K becomes important. In the FLO-2D model if K = 0 in the SED.DAT file, the value of K is automatically computed from the Manning's n-value.

Table 7. Resistance Parameters for Laminar Flow[1]

Surface	Range of K
Concrete/asphalt	24 -108
Bare sand	30 - 120
Graded surface	90 - 400
Bare clay - loam soil, eroded	100 - 500
Sparse vegetation	1,000 - 4,000
Short prairie grass	3,000 - 10,000
Bluegrass sod	7,000 - 50,000

[1] Woolhiser (1975)

The flow resistance n_{td} of the turbulent and dispersive shear stress components are combined into an equivalent Manning's n-value for the flow:

$$S_{td} = \frac{n_{td}^2 V^2}{h^{4/3}}$$

At very high concentrations, the dispersive stress arising from sediment particle contact increases the flow resistance n_{td} by transferring more momentum flux to the boundary. To estimate this increase in flow resistance, the conventional turbulent flow resistance n-value n_t is increased by an exponential function of the sediment concentration C_v:

$$n_{td} = n_t\ b\ e^{mCv}$$

where: n_t is the turbulent n-value, b is a coefficient (0.0538) and m is an exponent (6.0896). This equation was based on unpublished paper by Julien and O'Brien (1998) that relates the dispersive and

turbulent resistance in hyperconcentrated sediment flows as function of the ratio of the flow depth to the sediment grain size.

The friction slope components can then be combined in the following form:

$$S_f = \frac{\tau_y}{\gamma_m h} + \frac{K \eta V}{8 \gamma_m h^2} + \frac{n_{td}^2 V^2}{h^{4/3}}$$

A quadratic equation solution to the above friction slope equation has been formulated in the FLO-2D model to estimate the velocity for use in the momentum equation. The estimated velocity represents the flow velocity computed across the floodplain or channel element boundary using the average flow depth between the elements. Reasonable values of K and Manning's n-value can be assumed for the channel and overland flow resistance. The specific weight of the fluid matrix γ_m, yield stress τ_y and viscosity η vary principally with sediment concentration. Unless a rheological analysis of the mudflow site material is available, the following empirical relationships can be used to compute viscosity and yield stress:

$$\tau_y = \alpha_2 \; e^{\beta_2 C_v} \quad \text{and} \quad \eta = \alpha_1 \; e^{\beta_1 C_v}$$

where α_i and β_i are empirical coefficients defined by laboratory experiment (O'Brien and Julien, 1988). The viscosity (poises) and yield stress (dynes/cm^2) are shown to be functions of the volumetric sediment concentration C_v of silts, clays and in some cases, fine sands and do not include larger clastic material rafted along with the flow (Table 8 and Figures. 22 and 23).

Table 8. Yield Stress and Viscosity as a Function of Sediment Concentration

Source	$\tau_y = \alpha e^{\beta Cv}$ (dynes/cm^2)		$\eta = \alpha e^{\beta Cv}$ (poises)	
	α	β	α	β
Field Data				
Aspen Pit 1	0.181	25.7	0.0360	22.1
Aspen Pit 2	2.72	10.4	0.0538	14.5
Aspen Natural Soil	0.152	18.7	0.00136	28.4
Aspen Mine Fill	0.0473	21.1	0.128	12.0
Aspen Watershed	0.0383	19.6	0.000495	27.1
Aspen Mine Source Area	0.291	14.3	0.000201	33.1
Glenwood 1	0.0345	20.1	0.00283	23.0
Glenwood 2	0.0765	16.9	0.0648	6.20
Glenwood 3	0.000707	29.8	0.00632	19.9
Glenwood 4	0.00172	29.5	0.000602	33.1
Relationships Available from the Literature				
Iida (1938)*	-	-	0.0000373	36.6
Dai et al. (1980)	2.60	17.48	0.00750	14.39
Kang and Zhang (1980)	1.75	7.82	0.0405	8.29
Qian et al. (1980)	0.00136	21.2	-	-
	0.050	15.48	-	-
Chien and Ma (1958)	0.0588	19.1-32.7	-	-
Fei (1981)	0.166	25.6	-	-
	0.00470	22.2	-	-

*See O'Brien (1986) for the references.

Conversion: Shear Stress: 1 Pascal (PA) = 10 dynes/cm^2
Viscosity: 1 PAs = 10 dynes-sec/cm^2 = 10 poises

The viscosity of the fluid matrix is also a function of the percent and type of silts and clays and fluid temperature. Very viscous mudflows have high sediment concentrations and correspondingly high yield stresses and may result in laminar flow although laminar flows in nature are extremely rare. Less viscous flows (mud floods) are always turbulent.

For a mudflow event, the average sediment concentration generally ranges between 20% and 35% by volume with peak concentrations approaching 50% (Table 7 and Figure 22). Large flood events such as the 100-year flood may contain too much water to produce a viscous mudflow event. Smaller rainfall events such as the 10- or 25-year return period storm may have a greater propensity to create viscous mudflows. Most watersheds with a history of mudflow events and will gradually develop a sediment supply in the channel bed such that small storms may generate mudflow surges. Most rainfall induced mudflows follow a pattern of flood response. Initially clear water flows from the basin rainfall-runoff may arrive at the fan apex. This may be followed by a surge or frontal wave of mud and debris (40 to 50% concentration by volume). When the peak arrives, the average sediment concentration generally decreases to the range of 30 to 40% by volume. On the falling limb of the hydrograph, surges of higher sediment concentration may occur.

To simulate mudflows with the FLO-2D model, the MUD switch in the CONT.DAT must be turned on (MUD = 1) and the viscosity and yield stress variables in SED.DAT file must be specified. It is recommended that the viscosity and yield stress exponents and coefficients from Table 9 be selected for inclusion in the SED.DAT file. The field sample Glenwood 4, for example, creates a very viscous mudflow. A volumetric sediment concentration or a sediment volume must then be assigned to the water discharge for a timestep in the discretized inflow hydrograph in the INFLOW.DAT file. The inflow sediment volume may represent channel scour, bank erosion or hillslope failure. The incremental sediment volume is tracked through the routing simulation and reported as a total sediment volume in the summary volume conservation tables. This total sediment volume should be reviewed to determine if this is a reasonable sediment supply or yield from the watershed.

When routing the mud flood or mudflow over an alluvial fan or floodplain, the FLO-2D model preserves continuity for both the water and sediment. For every grid element and timestep, the change in the water and sediment volumes and the corresponding change in sediment concentration are computed. At the end of the simulation, the model reports on the amount of water and sediment removed from the study area (outflow) and the amount and location of the water and sediment remaining on the fan or in the channel (storage). The areal extent of mudflow inundation and the maximum flow depths and velocities are a function of the available sediment volume and concentration which can be varied in the FLO-2D simulations. For further discussion on model hyperconcentrated sediment flows, refer to the FLO-2D document *"Simulating Mudflows_Guidelines.doc"*.

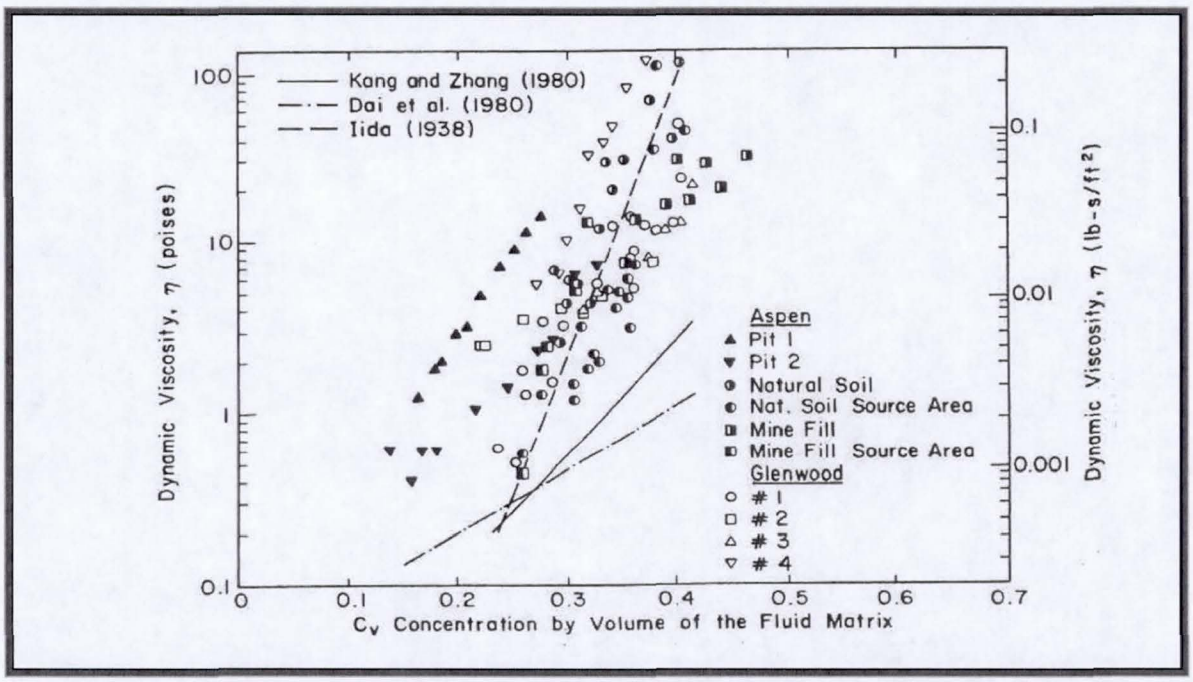


Figure 22. Dynamic Viscosity of Mudflow Samples versus Volumetric Concentration

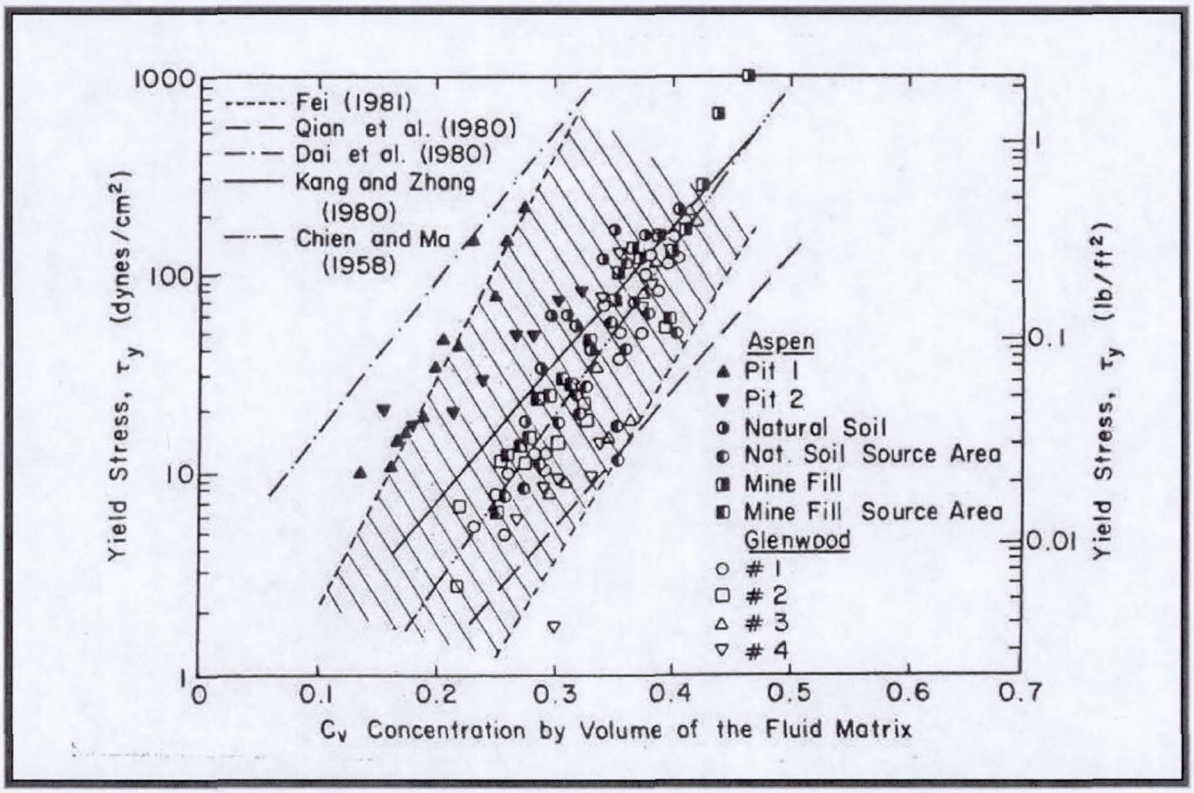


Figure 23. Yield Stress of Mudflow Samples versus Volumetric Concentration

V. FLO-2D APPLICATIONS AND METHODS

5.1 River Applications

Simulating river flow is one of the more common applications of the FLO-2D model (Figure 24). Abrupt cross section transitions, flat bed slopes, confluences and limited data bases are issues related to channel flow. The key to simulating river flooding is correctly assessing the relationship between the flood volume in the channel and the volume distributed on the floodplain. There are several considerations to defining channel volume and geometry. The surveyed channel cross sections should be appropriately spaced to model transitions between wide and narrow cross sections. The estimate of the total channel length (sum of the channel grid element lengths) is important to channel volume computation. Finally, surveyed water surface elevations at known discharges are needed to calibrate the channel roughness values. Channel routing with poorly matched channel geometry and estimated roughness can result in discharge surging.

When preparing a channel simulation, the available cross sections are distributed to the various channel elements based on reaches with similar geomorphic features. The bed elevation is then adjusted between channel elements with surveyed cross sections. The n-values are estimated from knowledge of the bed material, bed forms, vegetation or channel planform. The n-values may also serve to correct any mismatched channel flow area and slope. Roughness values can also be adjusted by specifying a maximum Froude number. Using this approach, the relationship between the channel flow area, bed slope and n-value can be adjusted to better represent the physical system, calibrate the water surface elevations, eliminate any numerical surging and speed-up the simulation.

The two most important FLO-2D results are the channel hydrograph at a downstream location and the floodplain area of inundation. Typically if the area of inundation is correct, then the channel flow depths and water surface elevations will be relatively accurate. Replicating the channel hydrograph and the floodplain inundation while conserving volume is a good indication that the volume distribution between the channel and the floodplain is reasonable.

Flood routing details related to channel flow include simulating hydraulic structures, levees, infiltration, sediment transport and hyperconcentrated sediment flows. Hydraulic structures may include bridges, culverts, weirs, diversions or any other channel hydraulic control. Levees are usually setback from the river on the floodplain, but can control the water surface in the channel if the flood is confined by the floodplain levees. Channel infiltration is based solely on the hydraulic conductivity and represents average bed and bank seepage conditions. Bed scour or deposition associated with a mobile analysis is non-uniformly distributed on the channel cross section. Finally, mudflows can be routed in channels.

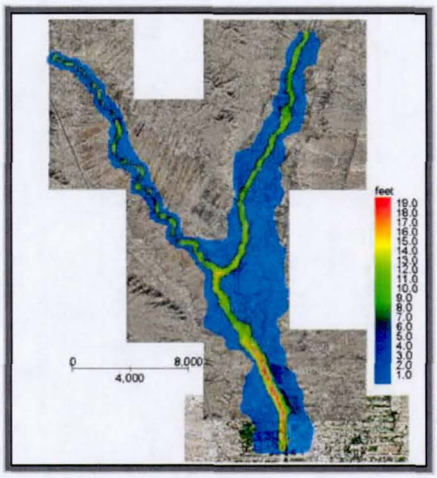


Figure 24. Middle Rio Grande and Rio Chama Confluence Model

5.2 Unconfined Overland and Alluvial Fan Floodimg

The primary focus of an unconfined flood simulation is how the volume is distributed over the floodplain surface. The flood volume controls the area of inundation (Figure 25). Important flood routing details include topography, spatial variation in infiltration and roughness, flow obstructions, levees, hydraulic structures and streets. The floodwave progression over the floodplain can be adjusted with the floodplain n-values. Street flow may control shallow flooding distribution in urban areas. Buildings and walls that obstruct flow paths and or eliminate floodplain storage (Figure 26). The levee routine can be used to simulate berms, elevated road fill, railroad embankments or other topographic features to confine the flow on the floodplain. Hydraulic conveyance facilities such as culverts, rainfall and gully flow may control the local water surface elevations.

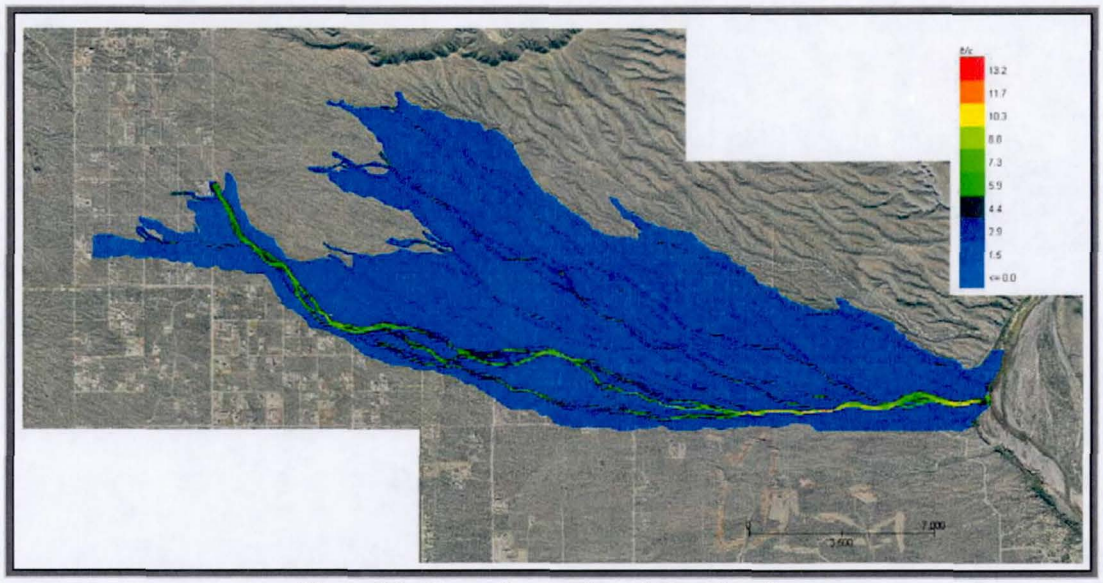


Figure 25. Unconfined Alluvial Fan Flooding

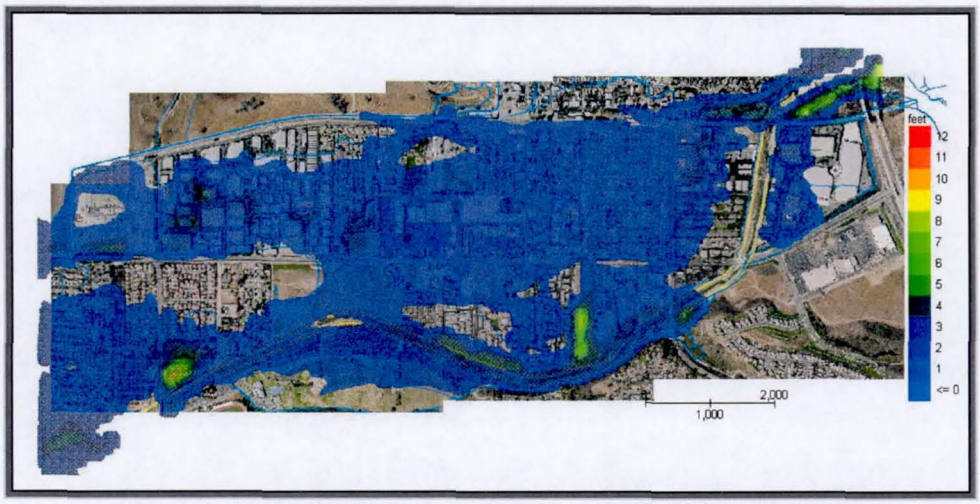


Figure 26. Urban flooding with Street Flow and Building Obstruction

FLO-2D can simulate an unconfined floodwave progression over a dry flow domain without specifying any boundary criteria. No hot starts or prescribed water surface elevations are required. It is possible to use the overland flow component to model various floodplain features such as detention basins, river channels or even streets. Flood retention basins have been modeled as part of the entire floodplain system using either the grid element elevation or levees to define the basin storage area. An appropriate grid element size should be selected to generate enough interior elements to adequately simulate the basin or channel. It should be noted that modeling the channel interior may require very small timesteps. Manning's open channel flow equation for the friction slope that is based on uniform, steady flow may not be appropriate if the water is ponded and the water surface is very flat.

If no inflow flood hydrograph data is available, FLO-2D can perform as a watershed model. Rainfall can occur on the floodplain surface resulting in sheet runoff after infiltration losses have been computed. It is possible to simulate rainfall while routing a flood event and have the rainfall occur on the inundated area. To improve concentration time, rill and gullies can be modeled to exchange flow between grid elements. This will reduce the travel time associated with sheet flow exchange between grid elements. Spatially variable rainfall distribution and a moving storm can be simulated. Real time rain gage data can also be modeled. The GDS will reformat the rain gage data for real time storm runoff and flood simulation.

Mud and debris flows can be simulated on alluvial fan surfaces. There are two methods for loading the hydrograph with sediment. A sediment concentration by volume is assigned to a discretized time interval of the inflow hydrograph. A second method is to load the inflow hydrograph with a volume of sediment. In this manner, spatially differential sediment loading in a watershed channel can be simulated. Once the hydrograph is bulked with sediment, the mudflow is routed as a water and sediment continuum over the hydrograph. The same water routing algorithm is used for mudflows but the momentum equation is solved with the additional viscous and yield stress terms. The bulked sediment hydrograph is tracked through system conserving volume for both water and sediment. Flow cessation and flow dilution are possible outcomes of the mudflow routing.

5.3 Model Results – What Constitutes a Successful Flood Simulation?

When a FLO-2D simulation is completed, how do you know if the simulation was successful or accurate? There are three keys to a successful project application:

- Volume conservation
- Area of inundation
- Maximum velocities and numerical surging.

Volume conservation must be conserved for both the overland flow and channel flow. If the volume was not conserved, then it will be necessary to conduct a more detailed review and determine where the volume conservation error occurred. If the volume was conserved, then the area of inundation can be quickly reviewed in either MAXPLOT or MAPPER programs. If the area of inundation seems reasonable and the flood appears to have progressed completely through the system, then the maximum velocities in the channel, on the floodplain or in the streets should be reviewed for discharge surging. By reviewing the results in MAXPLOT or MAPPER, the maximum floodplain velocities can be checked for unreasonably high velocities. The *Pocket Guide* and troubleshooting section in the Data Input Manual has more discussion on maximum velocities, numerical surging and how to resolve them including applications of the limiting Froude number.

Once the FLO-2D flood simulation is providing reasonable results, you can fine tune the model and speed it up. Review the TIME.OUT file to determine which channel, floodplain or street elements are causing the most timestep reductions. Model speed may not be critical if the simulation is accurate with respect to volume conservation, discharge surging and area of inundation.

VI. FLO-2D MODEL VALIDATION

The FLO-2D model has been applied on numerous projects by engineers and floodplain managers worldwide. Many users have performed validity tests including physical model studies. Users evaluate must whether the predicted hydraulics are reasonable and accurate for their projects. On occasion, a simulation may require some assistance to address a complex flow problem.

In January 1999, the Sacramento District Corps of Engineers conducted a review of several model applications, including a number of unconfined flood hazard projects, a California Aqueduct test case and replication model of the Arroyo Pasajero March 1995 alluvial fan flooding. Over a three-year period, the Corps actively engaged in model enhancement, code modification and model testing to expand the model applicability. In January 1999, the Sacramento District prepared a FLO-2D acceptance letter to FEMA. In early 2001, the Albuquerque District of the Corps of Engineers also completed a review of the FLO-2D model for riverine studies and submitted an acceptance letters to FEMA in support of using the FLO-2D for flood insurance studies. FLO-2D is on FEMA's list of approved hydraulic models for both riverine and overland flow (alluvial fan) flood studies.

Validation of hydraulic models with actual flood events is dependent on several factors including estimates of flow volume and area of inundation, appropriate estimates of flow resistance, representative conveyance geometry, accurate overland topography and measured flow hydraulics including water surface elevation, velocities and flow depths. The tools for validating hydraulic models include physical model (prototype) studies, comparison with other hydraulic numerical models or replication of past flood events. FLO-2D Software, Inc. maintains a series of validation tests. Ideally, the best model test involves the prediction of a flood event before it occurs; however, the probability of an actual flood having the similar volume to the predicted flood event is remote.

To confirm the accuracy of the FLO-2D model, several validation methods are maintained:

- Channel flow in the mild sloped California Aqueduct;
- Channel flow results compared to HEC-2 model results;
- Channel and floodplain flow routing for an actual river flood, Truckee River;
- Channel routing in a large river system (Green River) with a dam release floodwave;
- Comparison of floodplain inundation with mapped wetted acreage (Middle Rio Grande);
- Verification of mudflow hydraulics through replication of a know event (Rudd Creek);
- Flume discharge for steady, uniform flow using the overland flood routing component (compared with the analog results);
- Channel replication of measured river gaging discharge (Rio Grande, Figure 27).

In the last case, the replication of dam release discharge with highly unsteady flow 30 miles downstream reveals the robust nature of the solution algorithm. The results of these tests confirm that the FLO-2D computation algorithms are accurate for both channel and overland flood routing.

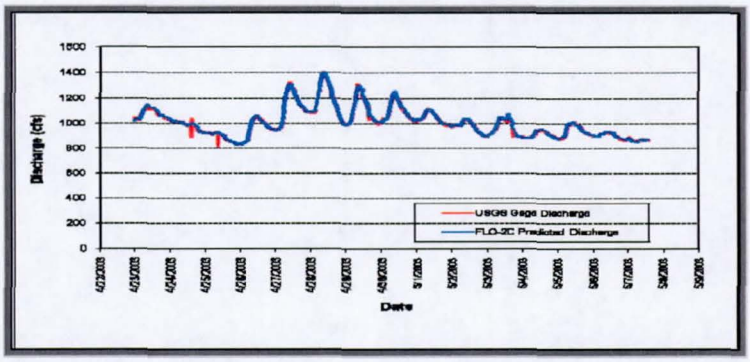


Figure 27. FLO-2D versus USGS Measured Gage Data

VII. REFERENCES

Ackers, P. and W.R. White, 1973. "Sediment Transport: New Approach and Analysis," J. of Hyd., ASCE, V. 99, no. HY11, pp. 2041-2060.

Bagnold, R.A., 1954. "Experiments on a gravity-free dispersion of large solid spheres in a Newtonian fluid under shear," Proc. of the Royal Society of London, Series A, V. 249, 235-297.

Chen, C.L., 1976. "Urban storm runoff inlet hydrograph study." Prepared for the Dept. of Transportation, PRWG 106-2, Utah State University, Logan Utah.

Cunge, J.A., F.M. Holly Jr., and A. Verwey, 1980. "Practical Aspects of Computational River Hydraulics," Pittman Advanced Publishing Program, London, UK.

DeLeon, A.A. and R.W. Jeppson, 1982. "Hydraulic and numerical solutions of steady-state, but spatially varied debris flow," Hydraulics and Hydrology Series, UWRL/H-82/03, Utah State Univ. at Logan, Utah.

Deng, Z., 1996. "Impact of Debris Flows and Its Mitigation," Ph.D. Dissertation submitted to the Dept. of Civil and Environmental Engineering, Univ. of Utah, Salt Lake City, Utah.

DMA, 1985. "Alluvial fan flooding methodology and analysis," Prepared for FEMA by DMA Consulting Engineers, Rey, California, October.

Egashira, S., K. Ashida, H. Yajima, and J. Takahama, 1989. "Constitutive equations of debris flow," Annuals of the Disaster Prevention Research Institute, Kyoto Univ., No. 32B-2, 487-501.

Einstein, H.A., 1950. "The bed-load function for sediment transportation in open channel flows," USDA Tech. Bull. No. 1026.

Engelund, F. and E. Hansen, 1967. "A monograph on sediment transport in alluvial streams," Teknisk Forlag, Copenhagen.

Fletcher, C.A.J., 1990. Computational Techniques for Fluid Dynamics, Volume I, 2nd ed., Springer-Velag, New York.

Fread, D.L., 1998. "Breach: An erosional model for earthern dam failures," National Weather Service, NOAA, Silver Spring, Maryland.

Fullerton, W.T., 1983. "Water and sediment routing from complex watersheds and example application to surface mining," Masters Thesis, Civil Engineering Dept., CSU, Fort Collins, CO.

Graf, W.H., 1971. Hydraulics of Sediment Transport. McGraw-Hill, New York, N.Y.

Green, W.H. and G.A. Ampt, 1911. "Studies on soil physics, part I: The flow of air and water through soils," J. of Agriculture Science.

Hashimoto, H., 1997. "A comparison between gravity flows of dry sand and sand-water mixtures," Recent Developments on Debris Flows, Springer, A. Armanini and M. Michiue (eds.), NY, NY.

Henderson, F. M., 1966. Open Channel Flow. MacMillan Publishing Co., Inc., NY, NY.

Hromadka, T.V. and C.C. Yen, 1987. "Diffusive hydrodynamic model," U. S. Geological Survey, Water Resources Investigations Report 87-4137, Denver Federal Center, Colorado.

James, W. P., J. Warinner, and M. Reedy, 1992. "Application of the Green-Ampt infiltration equation to watershed modeling," Water Resources Bulletin, AWRA, 28(3), pp. 623-634.

Jia, Y., 1990. "Minimum Froude Number and the Equilibrium of Alluvial Sand Rivers," Earth Surface Processes and Landforms, John Wiley & Sons, London, Vol. 15, 199-200.

Jin, M. and D.L. Fread, 1997. "Dynamic flood routing with explicit and implicit numerical solution schemes," J. of Hyd. Eng., ASCE, 123(3), 166-173.

Julien, P.Y., 1995. Erosion and Sedimentation. Cambridge University Press, New York, N.Y.

Julien, P.Y. and J.S. O'Brien, 1998. "Dispersive and turbulent stresses in hyperconcentrated sediment flows," Unpublished paper.

Julien, P.Y. and Y.Q. Lan, 1991. "On the rheology of hyperconcentrations", J. of Hyd. Eng., ASCE, 117(3), 346-353.

Karim, F., 1998. "Bed Material Discharge Prediction for Nonuniform Bed Sediments," J. of Hyd. Eng., ASCE, 124(6), 597-604.

Kusler, J., 1986. "Introduction: flooding in arid and semi-arid regions," Proc. of a Western State High Risk Flood Areas Symposium, 'Improving the Effectiveness of Floodplain Management in Arid and Semi-Arid Regions,' Association of State Floodplain Managers, Inc., Las Vegas, NV, March.

Laursen, E.M., 1958. "The total sediment load of streams," J. of the Hyd. Div., ASCE, V. 84, No HY1, 1530-1536.

MacArthur, R.C. and D.R. Schamber, 1986. "Numerical methods for simulating mudflows," Proc. of the 3rd Intl. Symp. on River Sedimentation, Univ. of Mississippi, 615-1623.

Major, J. and T.C. Pierson, 1992. "Debris flow rheology: experimental analysis of fine-grained slurries," Water Resources Research, 28(3), 841-857.

Meyer-Peter, E. and R. Muller, 1948. "Formulas for bedload transport," Proc. IAHR, 2nd Congress, Stockholm, 39-64.

Mussetter, R.A., P.F. Lagasse, M.D. Harvey, and C.A. Anderson, 1994. "Sediment and Erosion Design Guide." Prepared for the Albuquerque Metropolitan Arroyo Flood Control Authority by Resource Consultants & Engineers, Inc., Fort Collins, CO.

O'Brien, J.S., 1984. "1983 Yampa River Cobble Reach Morphology Investigation," Submitted to the U.S. Fish & Wildlife Service, Salt Lake City, Utah, Prepared by: Civil Engineering Dept., Colorado State University, Fort Collins, Colorado.

O'Brien, J.S., 1986. "Physical processes, rheology and modeling of mudflows," Doctoral dissertation, Colorado State University, Fort Collins, Colorado.

O'Brien, J.S. and P.Y. Julien, 1985. "Physical processes of hyperconcentrated sediment flows," Proc. of the ASCE Specialty Conf. on the Delineation of Landslides, Floods, and Debris Flow Hazards in Utah, Utah Water Research Laboratory, Series UWRL/g-85/03, 260-279.

O'Brien, J.S. and P.Y. Julien, 1987. Discussion on "Mountain torrent erosion," By K. Ashida in Sediment Transport in Gravel-Bed Rivers, John Wiley & Sons, 537-539.

O'Brien, J.S. and P.Y. Julien, 1988. "Laboratory analysis of mudflow properties," J. of Hyd. Eng., ASCE, 114(8), 877-887.

O'Brien, J.S., P.Y. Julien and W.T. Fullerton, 1993. "Two-dimensional water flood and mudflow simulation," J. of Hyd. Eng., ASCE, 119(2), 244-259.

O'Brien, J.S. and Simons, Li & Associates, Inc., 1989. "Flood hazard delineation for Cornet Creek, Telluride, Colorado," Submitted to Federal Emergency Management Agency, March.

Ponce, S.M., 1989. Engineering Hydrology, Prentice Hall, Englewood Cliffs, New Jersey.

Ponce, V.M., R.M. Li and D.B. Simons, 1978. "Applicability of kinematic and diffusion models," J. of Hyd. Div., ASCE, 104(1443), 353-60.

Ponce, V.M. and F.D. Theurer, 1982. "Accuracy Criteria in Diffusion Routing," J. of Hyd. Eng., ASCE, 108(6), 747-757.

Schamber, D.R. and R.C. MacArthur, 1985. "One-dimensional model for mudflows," Proc. of the ASCE Spec. Conf. on Hydraulics and Hydrology in the Small Computer Age, V. 2, 1334-39.

Simons, D.B. and F. Senturk, 1976. Sediment Transport Technology, Water Resource Publications, Fort Collins, CO.

Shen, H.W. and J.Y. Lu, 1983. "Development of predictions of bed armoring," J. of Hydraulic Engineeing, ASCE, 109(4), 611 - 629.

Smart, G.M., 1984. "Sediment Transport Formula for Steep Channels," J. of Hydraulic Engineering, ASCE, 110(3), 267-275.

Takahashi, T., 1979. "Debris flow on prismatic open channel flow," J. of the Hyd. Div., ASCE, 106(3), 381 - 396.

Takahashi, T. and H. Tsujimoto, 1985. "Delineation of the debris flow hazardous zone by a numerical simulation method," Proc. of the Intl. Symp. on Erosion, Debris Flow and Disaster Prevention, Tsukuba, Japan, 457-462.

Takahashi, T. and H. Nakagawa, 1989. "Debris flow hazard zone mapping," Proc. of the Japan - China (Taipai) Joint Seminar on Natural Hazard Mitigation, Kyoto, Japan, 363-372.

Toffaleti, F.B., 1969. "Definitive computations of sand discharge in rivers," J. of the Hyd. Div., ASCE, V. 95, no. HY1, pp. 225-246.

U.S. Army Corps of Engineers, 1988. "Mud Flow Modeling, One- and Two-Dimensional, Davis County, Utah," Draft report for the Omaha District, Omaha, NE, October.

U.S. Army Corps of Engineers, 1990. "HEC-1, Flood Hydrograph Package," User's Manual, Hydrologic Engineering Center, Davis, CA.

U.S. Army Corps of Engineers, 1990. "HEC-2 Water Surface Profiles," Hydrologic Engineering Center, Davis, CA.

U.S. Army Corps of Engineers, 1997. "Flood-Runoff Analysis," Technical Engineering and Design Guides, No. 19., ASCE Press, NY, NY.

Woo, H.S. P.Y. Julien, and E.V. Richardson, 1988. "Suspension of large concentrations of sand," J. Hyd. Eng., ASCE, 114, no. 8, pp. 888-898.

Woolhiser, D.A., 1975. "Simulation of Unsteady Overland Flow," in Unsteady Flow in Open Channels, Mahmood, K. and Yevjevich, V. eds., Water Resources Publications, Fort Collins, CO.

Yang, C.T., 1973. "Incipient motion and sediment transport," J. of Hyd Div. ASCE, V. 99, no. HY10, pp. 1679-1704.

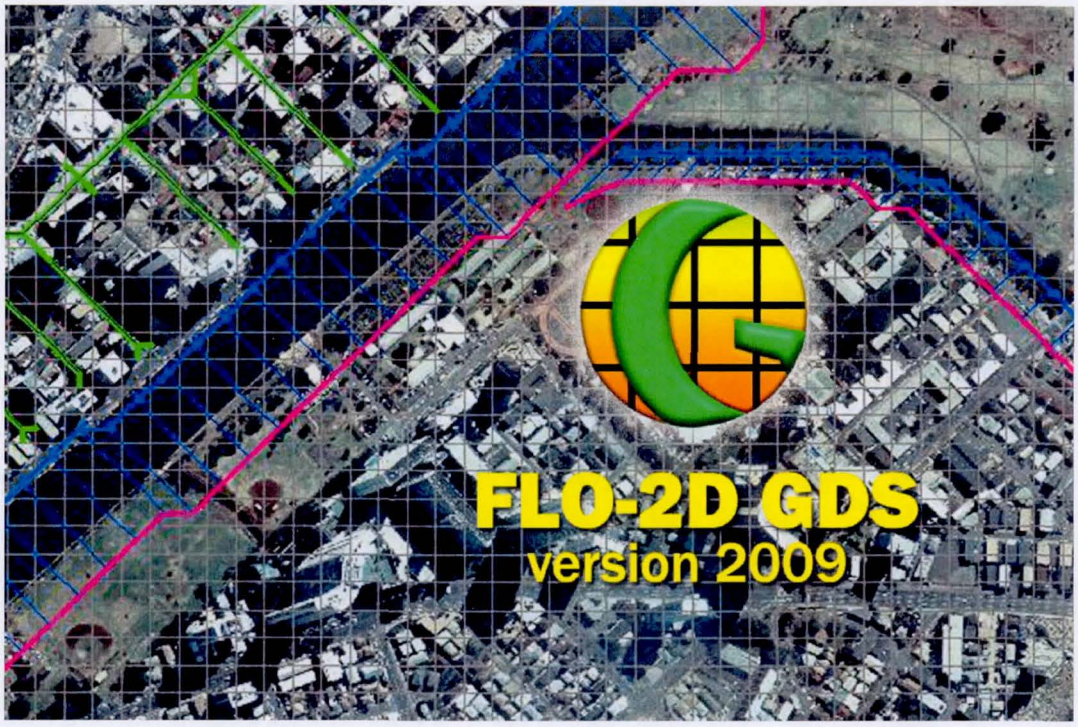
Yang, C.T., 1996. Sediment Transport, Theory and Practice," McGraw-Hill, New York, N.Y.

Zeller, M. E. and W. T. Fullerton, 1983. "A Theoretically Derived Sediment Transport Equation for Sand-Bed Channels in Arid Regions," Proceedings of the D. B. Simons Symposium on Erosion and Sedimentation, R. M. Li and P. F. Lagasse, eds., Colorado State University and ASCE.

FLO-2D

Grid Developers System
GDS
Users Manual

Version 2009.06



May 2009

Table of Contents

Page

I. Introduction 1

1.1 What's New in GDS 2009 3

II. Data Requirements 5

III. Getting Started 6

3.1 Create a Grid System by Digitizing Points 8

3.2 Create a Grid System from DTM Points 10

3.3 Create a Grid System from ArcInfo ASCII Grid File 11

3.4 Create a Grid System from an Existing Shape File 13

3.5 Import an Existing *FLO-2D* Project 14

3.6 Start a new project using a CAD file 16

3.7 Start a new project from a *HEC-RAS* project file 16

3.8 Grid System Generation 17

3.9 Assigning Manning's Roughness Coefficients (n-values) 22

3.10 Interpolation of Terrain Elevations 22

IV. Mouse Button Commands 26

4.1 Left Button actions (one-click) 26

4.2 Left Button actions (double-click) 27

4.3 Right Button actions 27

V. Toolbar and Menus 29

5.1 Toolbar ..... **Error! Bookmark not defined.**

5.2 File Menu Commands: 30

5.2.1 Define Working Region Command (File Menu) 30

5.2.2 New Project/from DTM Elevation Points Command (File Menu) 31

5.2.3 New Project/from Existing ArcInfo ASCII Grid File... Command (File Menu) 32

5.2.4 New Project/from Existing Shapefile…Command (File Menu) 33

5.2.5 New Project/from *FLO-2D* Project…Command (File Menu) 33

5.2.6 New Project/from Existing CAD File Command (File Menu) 35

5.2.7 New Project/from Existing *HEC-RAS* .PRJ File Command (File Menu) 36

5.2.8 Open .TOP Project… Command (File Menu) 36

5.2.9 Open Existing *FLO-2D* Project … Command (File Menu) 37

5.2.10 Save .TOP Project… Command (File Menu) 37

5.2.11 Save *FLO-2D* Files Command (File Menu) 38

5.2.12 Run *FLO-2D*… Command (File Menu) 38

5.2.13 Run Mapper Command (File Menu) 40

5.2.14 Run PROFILES Command (File Menu) 40

5.2.15 Run RAIN Command (File Menu) 41

5.2.16 Create FPLAIN.DAT and CADPTS.DAT Files Command (File Menu) 41

5.2.17 Import Image/Individual Image Command (File Menu) 42

5.2.18 Import Image/Group of Images Command (File Menu) 44

5.2.19 Import Elevation Points/ DTM Points… Command (File Menu) 45

5.2.20 Individual ArcInfo ASCII Grid File Command (File Menu) 46
5.2.21 ArcInfo ASCII Grid File Catalog Command (File Menu) 47
5.2.22 Import Shape File Command (File Menu) 48
5.2.23 Import ArcInfo ASCII Grid File (File Menu) 49
5.2.24 Import *HEC-RAS* Channels…(File Menu) 49
5.2.25 Import CAD Graphic Elements…(File Menu) 50
5.2.26 Save Elevation Points (File Menu) 50
5.2.27 Export…Command (File Menu) 50
5.2.28 Exit Command (File Menu) 52
5.3 View Menu Commands: 53
5.3.1 View All Command (View Menu) 53
5.3.2 Zoom Out Previous View Command (View Menu) 54
5.3.3 Zoom Out 10% Command (View Menu) 54
5.3.4 Pan Command (View Menu) 55
5.3.5 Layers List Command (View Menu) 55
5.3.6 DTM Point Elevations Rendering Command (View Menu) 57
5.3.7 Track Elevation Points Command (View Menu) 58
5.3.8 Grid Element Numbers, Elevations, and n-values and Curve Numbers Commands (View Menu) 59
5.3.9 Grid...Command (View Menu) 60
5.3.10 View Components Commands (View Menu) 61
5.3.11 View Elevation Points Commands (View Menu) 63
5.3.12 Manning's n Values Rendering Command (View Menu) 63
5.3.13 Grid Element Elevation Rendering Command (View Menu) 64
5.3.14 Highlight Grid Element Number… (View Menu) 64
5.3.15 Cross Section Numbers… (View Menu) 65
5.3.16 Cells Without Cross Section Numbers… (View Menu) 66
5.3.17 Redraw Command (View Menu) 66
5.4 Design Menu Commands: 67
5.4.1 Elevation Points/Insert Command (Design Menu) 67
5.4.2 Elevation Points/Create Elevation Points Layer Command (Design Menu) 67
5.4.3 Elevation Points/Delete Elevation Points from Selected Area Command (Design Menu) ... 68
5.4.4 Grid Element Text Style Commands (Design Menu) 69
5.4.5 Channel Style (Design Menu) 70
5.4.6 Area Reduction Factor Style (Design Menu) 70
5.5 Grid Menu Commands: 71
5.5.1 Create Grid Command (Grid Menu) 71
5.5.2 Select/Grid Element Command (Grid Menu) 72
5.5.3 Select/Grid Elements Defined by Polygon Command (Grid Menu) 72
5.5.4 Select/Inner Grid Elements Command (Grid Menu) 73
5.5.5 Select/Unselect All Command (Grid Menu) 73
5.5.6 Assign Parameters to Selection/Elevations Command (Grid Menu) 73
5.5.7 Assign Parameters to Selection/Manning's Coefficient Command (Grid Menu) 74
5.5.8 Assign Parameters to Selection/Area and With Reduction Factors Command (Grid Menu) 75
5.5.9 Assign Parameters to Selection/Levee Command (Grid Menu) 77

5.5.10 Assign Parameters to Selection/Multiple Channels Command (Grid Menu) 80
5.5.11 Assign Parameters to Selection/Inflow/Outflow Condition Command (Grid Menu)..... 82
5.5.12 Assign Parameters to Selection/Infiltration Command (Grid Menu) 88
5.5.13 Assign Parameters to Selection/No Discharge Exchange Command (Grid Menu)..... 89
5.5.14 Assign Parameters to Selection/Time-Variant Groundwater Head Command (Grid Menu) 89
5.5.15 Assign Parameters to Selection/Rigid Bed Element Command (Grid Menu) 90
5.5.16 Interpolate Elevation Points Command (Grid Menu)..... 90
5.5.17 Compute Green-Ampt Parameters Command (Grid Menu)..... 91
5.5.18 Compute Manning Coefficients… Command (Grid Menu)..... 97
5.5.19 Compute SCS Curve Number … Command (Grid Menu)..... 98
5.5.20 Define Boundary Grid Elements/Element by Element Command (Grid Menu) 106
5.5.21 Define Boundary Grid Elements/ Draw Line Command (Grid Menu) 106
5.5.22 Setup Computational Area/Click Inside Modeling Area Domain Command (Grid Menu) 108
5.5.23 Setup Computational Area/ Define Modeling Boundary with Polygon Command (Grid Menu) 108
5.5.24 Create Grid Layer Command (Grid Menu) 109
5.6 Tools Menu Commands: 109
5.6.1 Options…/Interpolation… Command (Tools Menu) 110
5.6.2 Options…/Change Refresh Count Command (Tools Menu) 112
5.6.3 Options…/Directory Paths Command (Tools Menu)..... 113
5.6.4 Measure Distance along Line Command (Tools Menu)..... 114
5.6.5 Compute Average Point Rainfall Depth Command (Tools Menu) 115
5.6.6 Interpolate Variable Rainfall Depth Command (Tools Menu)..... 117
5.6.7 Levee Express Editor Command (Tools Menu)..... 119
5.6.8 Floating Points Variables Express Editor Command (Tools Menu) 120
5.6.9 Create Channel Segment Command (Tools Menu)..... 121
5.6.10 Create Channel Segment with a Polyline Command (Tools Menu)..... 127
5.6.11 Create Cross Section Command (Tools Menu)..... 128
5.6.12 Create Cross Section File from Existing N-shape Channel Cells Command (Tools Menu) 128
5.6.13 Create Floodplain Cross Sections Command (Tools Menu) 129
5.6.14 Create Street Segment Command (Tools Menu)..... 131
5.6.15 Create Street Segment with a Polyline Command (Tools Menu)..... 134
5.6.16 Create Detention Basin Command (Tools Menu) 135
5.6.17 Mud and Sediment Transport Command (Tools Menu)..... 136
5.6.18 Evaporation Command (Tools Menu)..... 137
5.6.19 *MODFLO-2D* Simulation Command (Tools Menu) 138
5.6.20 Hydraulic Structures Command (Tools Menu) 139
5.6.21 Rain Command (Tools Menu)..... 141
5.6.22 Breach Command (Tools Menu)..... 142
Global Breach Data 143
Levee Fragility Curve Data 144
Individual Breach Data 145

5.6.23 NOFLOCS Command (Tools Menu) 146
5.6.24 Find Watershed Command (Tools Menu) 147
5.6.25 Convert *HEC-RAS* Xsec to *FLO-2D* Command (Tools Menu) 147
5.6.26 Convert *HEC-RAS* Channels to *FLO-2D* Channel Command (Tools Menu) 148
5.6.27 Levee Profile Command (Tools Menu) 148
5.6.28 Street Profile Command (Tools Menu) 150
5.7 Help Command: 151

I. Introduction

The *FLO-2D* Grid Developer System (*GDS*) is a GIS integrated software tool used to facilitate the creation of all required data to run the *FLO-2D* model. GDS simplifies construction of the finite difference grid system and allows graphical editing of its attributes and components. The *GDS* will overlay the grid system on a digital terrain map or a Digital Terrain Model (DTM) set of points and will interpolate and assign elevations to the grid elements. It will establish geographical boundaries, import background images and photos, and it will automatically generate the data files required to run the model. These files are generated and edited in an 'error free' format that requires no further modification to run the *FLO-2D* model.

GDS can also run the *FLO-2D* model once the data files have been created. One of primary functions of the *GDS* is to graphically edit grid elements attributes such as streets, flow obstructions, levees, rainfall, evaporation, infiltration, channels, hydraulic structures, sediment transport and mud flow parameters, detention basins, and n-values interactively using mouse point and click events. All data is validated to minimize common input errors. For GIS integration *GDS* makes extensive use of ESRI MapObjects© Version 2.4 GIS software controls.

The *GDS* system can perform the following functions:

Interactive Data Entry

- Interactive graphical editing of channels, streets, multiple channels, levees and buildings and other grid element attributes.
- Input detention basins parameters.
- Assign attributes to group of user selected grid elements
- Assign reservoir or pounded water elevation for an initial condition on a portion of the grid system.
- Assign channel NOFLOC values (channel elements not sharing discharge) including NOFLOCs associated with opposite bank tributaries due to channel extensions.
- Assign spatially and temporally variable rainfall data.
- Edit levee attributes using the Levee Express Editor.

Import Data

- Import HEC-1 hydrographs.
- Import DTM points in ASCII format.

Data Processing

- Automatic generation of finite difference grid and selection of computation area from user defined polygons.
- Compute spatially variable Green-Ampt infiltration parameters using soil and land use shape files.
- Compute spatially variable Manning's n-values using shape files.
- Convert *HEC-RAS* cross section data to *FLO-2D* channel cross section data format.

- Compute lengths for channel segments.
- Create the files to run a *FLO-2D* simulation
    - CONT.DAT
    - TOLER.DAT
    - INFLOW.DAT
    - OUTFLOW.DAT
    - RAIN.DAT
    - INFIL.DAT
    - EVAPOR.DAT
    - CHAN.DAT
    - CHANBANK.DAT
    - XSEC.DAT
    - HYSTRUC.DAT
    - STREET.DAT
    - ARF.DAT
    - MULT.DAT
    - SED.DAT
    - LEVEE.DAT
    - FPXSEC.DAT
    - BREACH.DAT
    - FPLAIN.DAT and
    - CADPTS.DAT.

Data Display and Visualization

- Show channel extension directions.
- Display interpolated elevation map.
- Display n-Manning's map.
- Display street and levee profiles.
- Display grid element numbers, elevations and Manning's n coefficients on the grid system.
- Zoom and pan.

GIS integration

- Import ESRI shape file format data such as land use, soil types, and Manning roughness coefficients.
- Import ESRI ArcInfo ASCII grid files containing terrain elevations and NOAA rainfall data.
- Import multiple geo-referenced aerial photos in various graphic formats such as TIFF, BMP, JPG, and others as background to flooding results.
- Customize multiple layer display and layer properties.

1.1 What's New in GDS 2009

GDS version 2009 includes a number of enhancements including:

Creation of new projects

- From *HEC-RAS* Project Files.
- From DXF and DWG CAD files.
- From ArcGIS Shape files.
- User may now change grid top left and bottom right coordinates.

New Import file options

- DXF and DWG CAD files.
- *HEC-RAS* geometry files.

Channel enhancements

- Create channel segments from *HEC-RAS* geometry files.
- Create channel segment from polyline.
- Create new cross sections and assign them to natural channel elements.
- Assign cross section number to a natural channel element.
- Create cross section data file (XSEC.DAT).
- Realign existing channel.
- Display cross section numbers associated to each natural channel element.
- Display grid elements for natural channel elements that do not have associated cross sections.
- Link to PROFILES cross section and slope interpolation program.

New Input dialogs

- Input sediment transport formulas and parameters.
- Input concentration or volume mud hydrographs.
- Input mud flow parameters.
- Input hydraulic structure data.
- Input rainfall and evaporation data.
- Input levee breach parameters.

New Tools

- Automatic and manual assignment of no exchange channel elements.
- Floating variables Express Editor: Interactive edition of area reduction factors, infiltration

and multiple channels.

- Select rigid bed elements for sediment transport analysis.
- Approximate watershed delineation.
- Outflow diversion (hydrograph) boundary condition.
- Link with USGS-*MODFLOW* groundwater flow model for integrated surface/subsurface simulations.

Visualization

- Rendering of terrain elevations as solid colors or hatched colored patterns.
- Display watershed and floodplain watercourse directions.
- Plot cross section for user selected natural channel elements.
- Plot 3D view of channel.

Validations

- Improved checks on data validity.

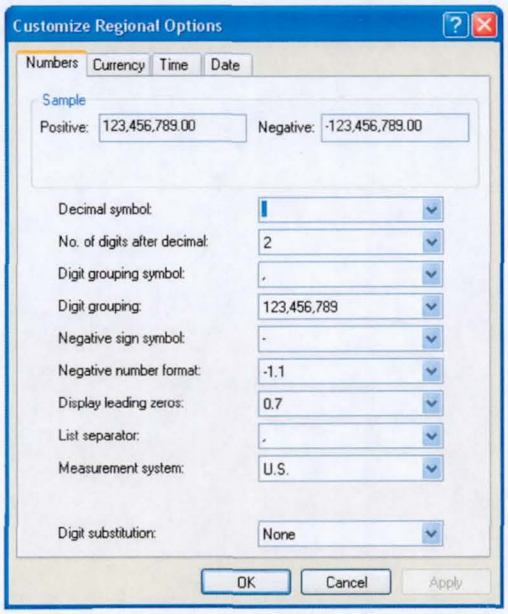
There are a number of *GDS* tutorials with examples to help you learn the various commands, features and tools. These tutorials are available on the *FLO-2D* installation CD. There are also example projects provided on the CD to work through with the tutorials.

II. Data Requirements

GDS data are introduced through window dialog boxes and are stored in files for later retrieval. User input data is validated to avoid out-of-range values. Default or recommended values are frequently displayed when the dialog boxes open. The *GDS* may require or use the following data:

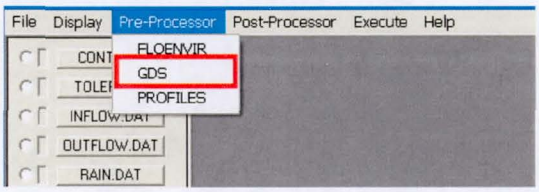
- Terrain elevation data represented as random topographic points (DTM);
- Terrain elevation data in ASCII grid files;
- Study region limits (coordinates);
- Manning's roughness (n-value) shape files;
- Soil shape files and tables;
- Land use shape files and tables;
- Image files in any of the following format: BMP, JPG, ArcInfo INFO Grid, GeoTIFF TIFF with a Geo header, Image Catalogs, JPEG, MrSID, TIFF, JPEG, BMP;
- Rainfall gage data in ASCII grid files;
- Flood inflow hydrographs.

IMPORTANT NOTE: *For the GDS system to function properly, the MS-Windows environment decimal symbol must be set to decimal point "." And the digit grouping symbol to comma ",". Set these options in the Control Panel/Regional and Language Options/Regional Options/Customize/ dialog box:*

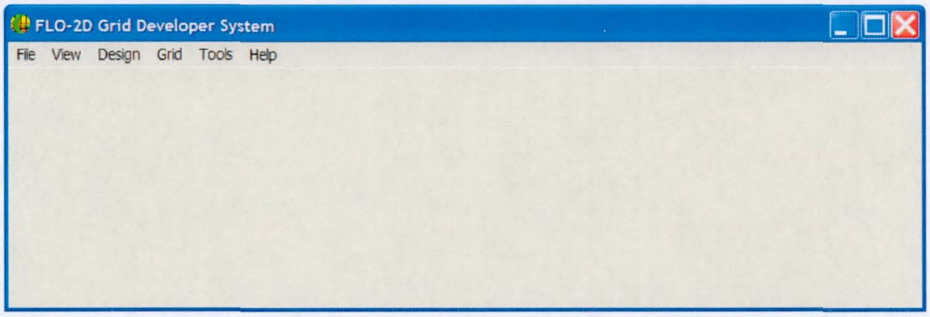


III. Getting Started

All *FLO-2D* programs can be run from Explorer by simply double clicking on the program executable name. You can also run the *GDS* from the *FLO-2D* GUI main window using the *GDS* command in the Post Processor menu:



GDS main window will appear:



To create a *FLO-2D* grid system, the following steps need to be completed:

1. Input the DTM points (terrain data) or interactively generate topographic contours;
2. Input the grid element size to create the base grid system;
3. Design the project area boundary polygon;
4. Interpolate the digital terrain elevation data to compute grid element elevations;
5. Set Manning's n-values;
6. Assign inflow hydrographs to selected grid elements;
7. Select outflow grid elements;
8. Set control and stability parameters;
9. Run the *FLO-2D* model.

A partial list of optional tasks include:

1. Spatially edit the Manning's n-values;
2. Create streets, buildings, levees and channels and assign their attributes;
3. Calculate and spatially varied Green-Ampt infiltration parameters;
4. Calculate variable rainfall and update the RAIN.DAT files;
5. Input detention basin locations and parameters.

There are five options to create a *FLO-2D* grid system with the *GDS* program. These options are provided by the *New Project*...Command in the File Menu:

Only one of these options is necessary to start a *GDS* session to build or edit a *FLO-2D* model:

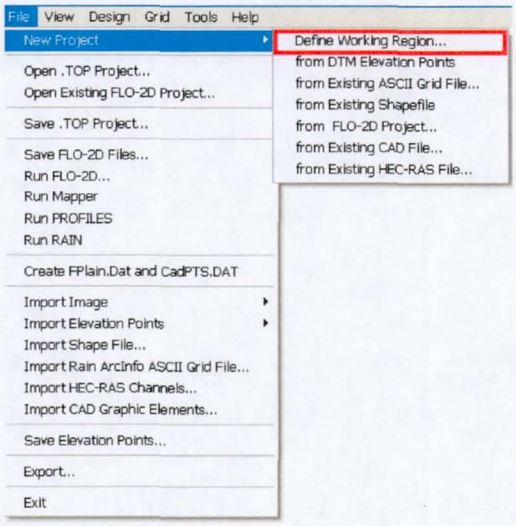
1. Create regional coordinate limits and add random point elevations;
2. Import DTM points (random coordinates and elevations) from an existing file;
3. Import an ArcInfo ASCII grid file containing terrain elevation in a regular grid;
4. Import a shape file containing modeling region extents.
5. Import an existing *FLO-2D* project (from *DAT files).

These options will be described in the following sections.

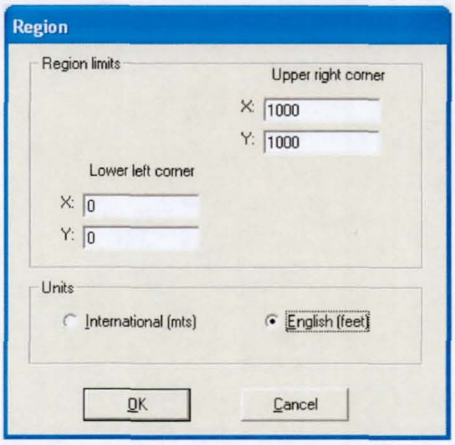
IMPORTANT NOTE: You can only have one project open at a time. If there is a project already open when you select this command, GDS prompts you to save the current project. However, you can have more than one GDS application open at a time.

3.1 Create a Grid System by Digitizing Points

This method is analogous to digitizing a map in a CADD program. First you input the potential flow domain coordinate limits and *GDS* establishes the screen working area. On the *File* menu, point to *New Project* and click *Define working region.*



Enter the coordinates of the upper right corner and the lower left corner in the dialog box:

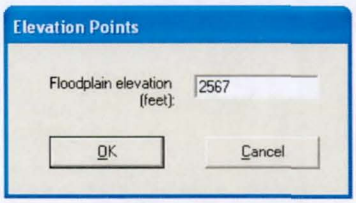


The values entered in the dialog box indicate that the lower left hand corner corresponds to the coordinates (0,0) feet and the upper right hand corner represents the coordinates (1000, 1000) feet. In this project the English System (ft) was selected for the coordinate units. The International System (meters) may be selected instead.

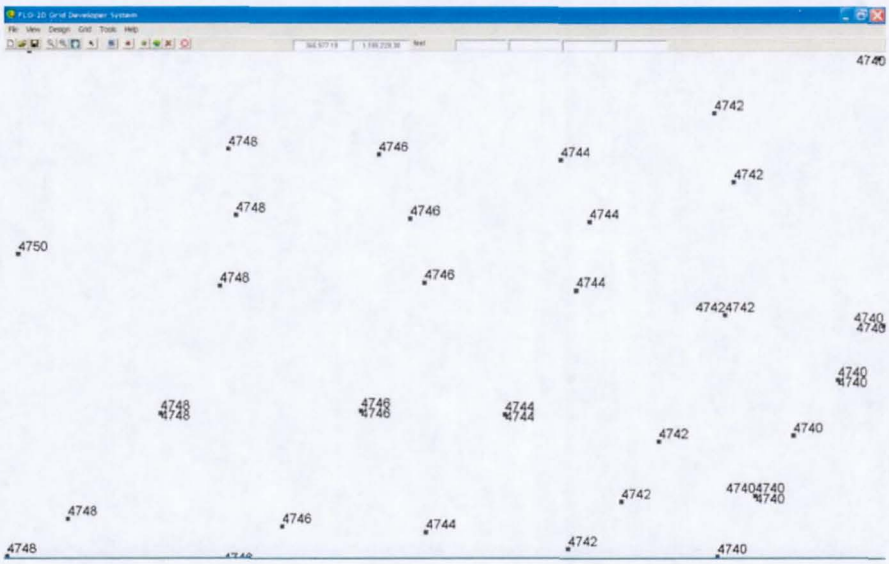
IMPORTANT NOTE: This selection will define the unit system that FLO-2D will use for all its data.

When you are finished entering the coordinates, click the *'OK'* button. Random

elevation points can now be added to the working area. In the toolbar just below the main menu is a set of icons that allows quick access to frequently used commands. To insert a new elevation point, click on the toolbar button or select the Elevation Point/*Insert* command from the Design menu. A dialog box will appear to enter the point elevation in the working area:



Press *'OK'* and then use the mouse to click on the point(s) corresponding to this elevation within the flow domain area. Every time the mouse button is clicked the point is marked on the screen as shown bellow. Repeat this process for all points with a similar elevation.



Please note that the boxes at the right side of Toolbar will indicate the X and Y coordinates of the cursor location in the working area, the Grid Element Index, the Grid Element Number, the Grid Element Interpolated Elevation and the Elevation of the DTM point. You can use a shape file or a scanned contour map as a background reference image to digitize contour lines. The *GDS* does not recognize the contour lines until the DTM point data is entered. The elevations are assigned by following the contour lines and clicking with the mouse to set a point elevation at appropriate intervals similar to digitizing contour lines in a CADD program.

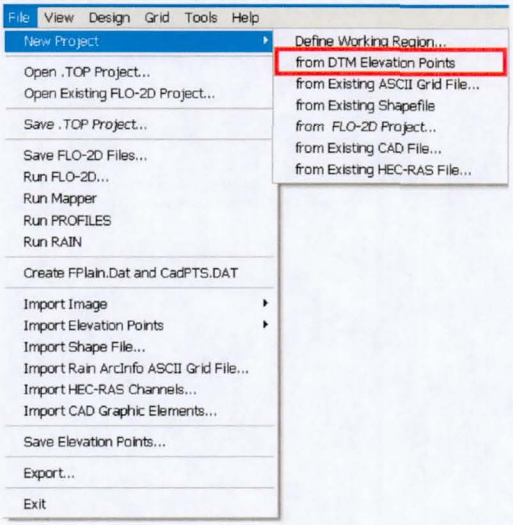
3.2 Create a Grid System from DTM Points

The second option to start a new project is to import an existing file of DTM points. This is the most common method for starting a *FLO-2D* project. The DTM data should be provided in ASCII file format (separated by a space or comma) listing the coordinates and elevation of each point:

```
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
```

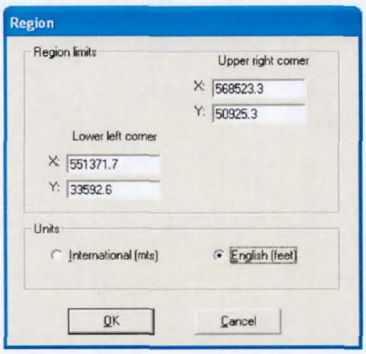
The first two columns are the x- and y- coordinates and the third column is elevation (ft or m). There can be an additional fourth column to accommodate a common format used by LiDAR data, but its values will be ignored by the *GDS*.

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

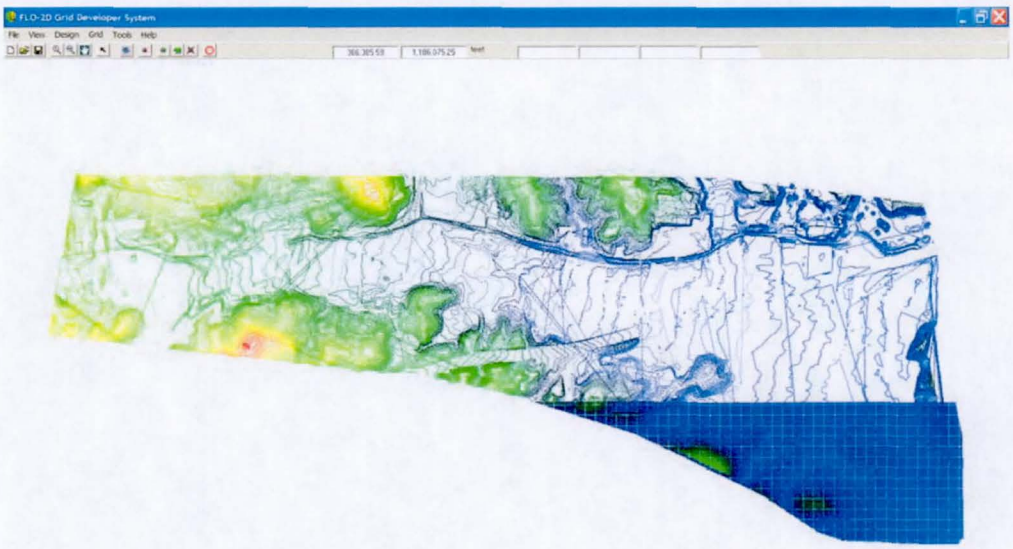


GDS searches for a default filename with *.PTS extension, but you may use files with other extensions provided that the file is in the appropriate format. Once the points are read from the file, they will be shown on the screen and the modeling region will be automatically scaled from the minimum and maximum coordinates.

Important Note: DTM point files with tabs, non-ASCII characters or end of file delimiters will cause an error in reading the file.



Chose the system of units (e.g. English) and click *'OK'*. The DTM points will be displayed:

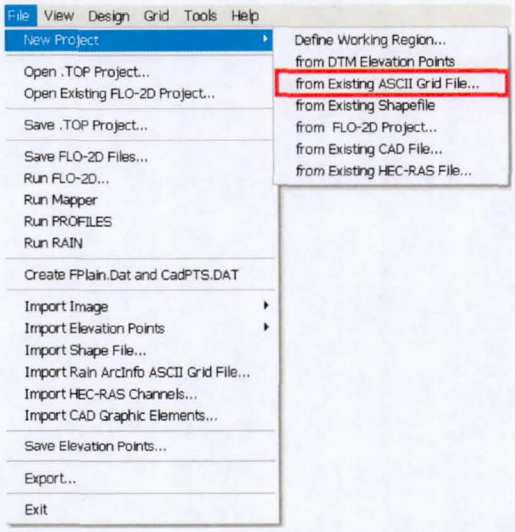


After the DTM points have been imported, the grid system can be overlaid and the computational domain area can be created as explained later.

IMPORTANT NOTE: At the present time, there is a maximum DTM point limitation of approximately 12 to 14 million points. For DTM point files larger than this, the grid system will have to be developed in segments and pasted together.

3.3 Create a Grid System from ArcInfo ASCII Grid File

A new project can be created by importing the terrain elevation points from an existing ArcInfo ASCII grid file. The working area is automatically scaled from the maximum and minimum point coordinates. To import the ArcInfo ASCII Grid File (*.ASC) file click on this command in the *File* menu:



Chose the appropriate filename in your project subdirectory. The format of these files is as follows:

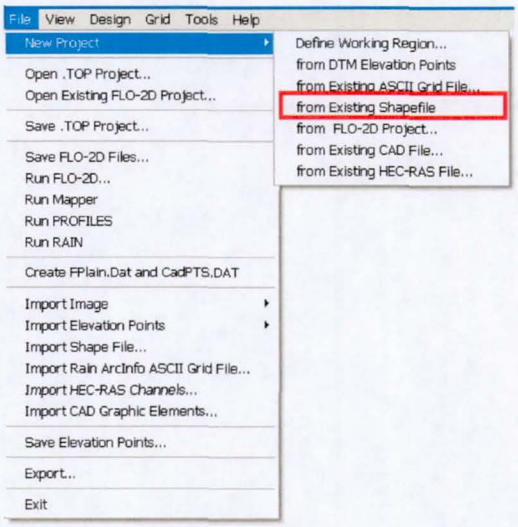
ncols /* Number of columns in the grid */
nrows /* Number of rows in the grid */
xllcorner x /* Lower left x coordinate of grid */
yllcorner y /* Lower left x coordinate of grid */
cellsize size /* Grid cell size */
NODATRA_value NODATA /* value of an empty grid cell */
z11 z12 z13 ... z1ncols /* values of row 1 */
z21 z22 z23 ... z2ncols /* values of row 2 */
…
…
znrows1 znrows2 znrows3 ... znrowsncols /* values of last row*/

The rows of data are read from north to south. For example:

ncols 388
nrows 461
xllcorner 674070.85270015
yllcorner 1000118.1562353
cellsize 100
NODATA_value -9999
2477.259 2480.868 2486.877 2486.877 2487.308 2490.641 2493.438…

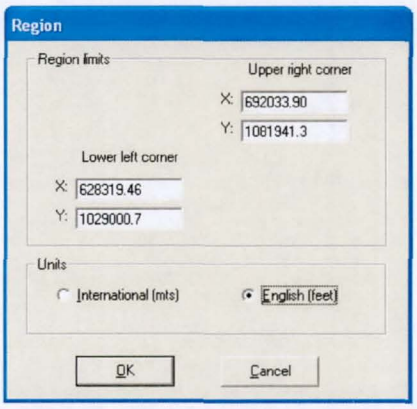
3.4 Create a Grid System from an Existing Shape File

A new project can be created by importing existing ESRI *point* Shape file. First click on *File/New Project/from Existing Shape file*:

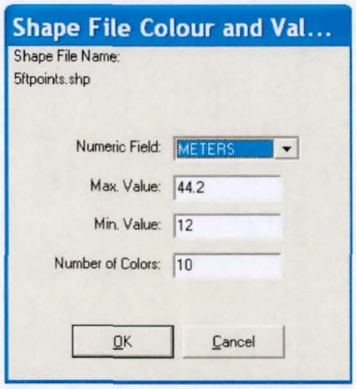


After selecting the appropriate POINTZ shape file name in your project subdirectory, the working region is automatically scaled from the existing project coordinates and the DTM elevation points are extracted from the shape files attributes.

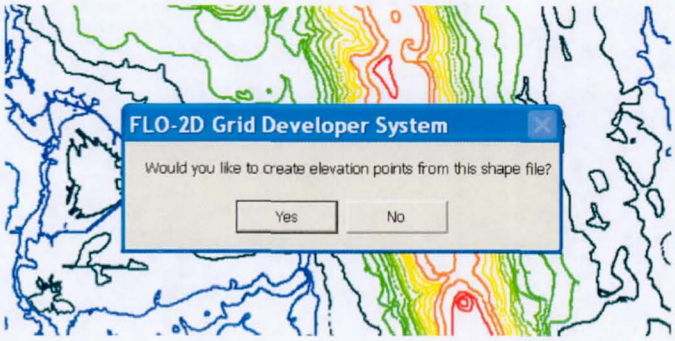
IMPORTANT NOTE: GDS is only able to extract data from POINTZ shape files, not from polygon or line shape files.



Chose the unit system (in this case English) and click *'OK'*. Then select the numeric field corresponding to the elevation attributes in the shape file and dialog box displays the elevation limits.



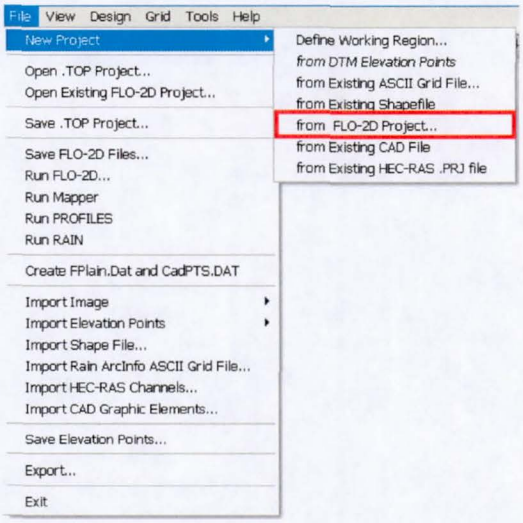
Click *'OK'* to accept the displayed values and the shape file elevations will be displayed. A message asks if you want to create the elevation layer necessary to obtain the DTM data from the shape file:



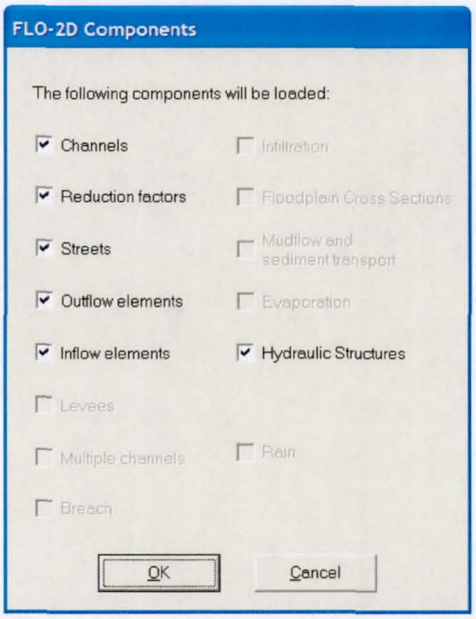
Click 'Yes' to generate a DTM points layer for editing and interpolating the DTM data.

3.5 Import an Existing *FLO-2D* Project

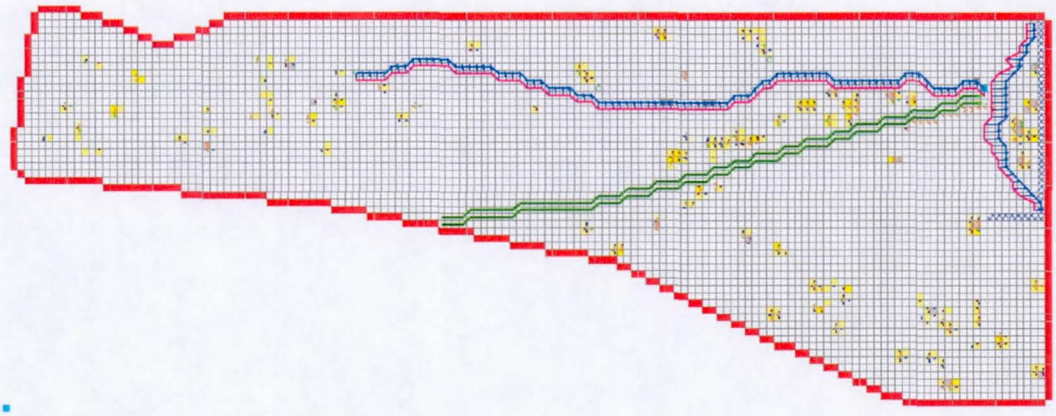
This option provides a way to start a *GDS* project by importing existing *FLO-2D* data files. The working region and *FLO-2D* grid system are automatically scaled from the existing project coordinates. To use this option, click *New Project/fromFLO-2D project* command in the *File* menu:



Select the FPLAIN.DAT file from your project subdirectory to open the project. Other data files corresponding to the *FLO-2D* components such as CONT.DAT, TOLER.DAT, STREETS.DAT, LEVEES.DAT, etc., will be detected and the following dialog box will allow the user to load selected components:



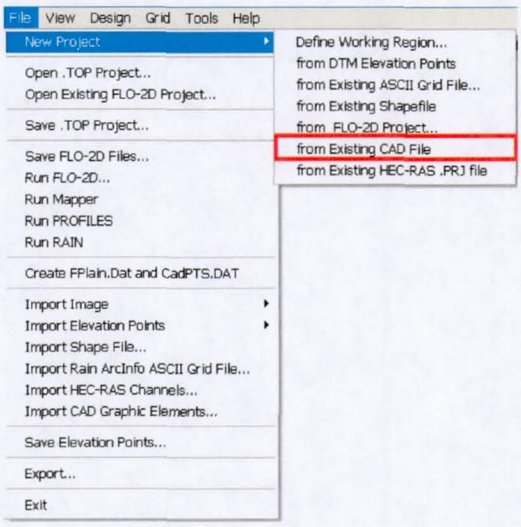
Check the available boxes to select or unselect the components to import all the data:



Select this project option when most of the *FLO-2D* files are already created you need to graphically edit the grid element attributes.

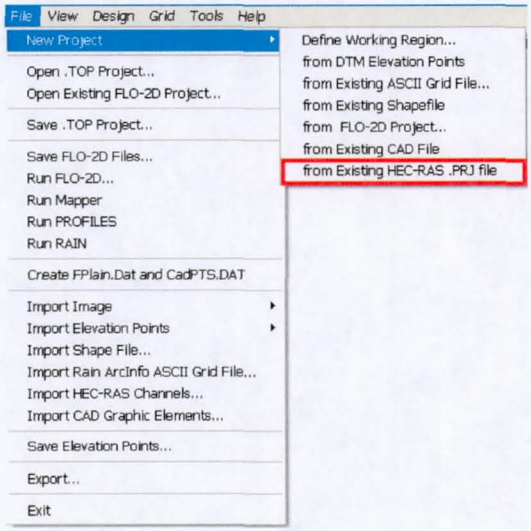
3.6 Start a new project using a CAD file

This option provides a way to start a *GDS* project by importing a DXF or DWG CAD files. The working region and *FLO-2D* grid system are automatically scaled from the existing CAD file extents.



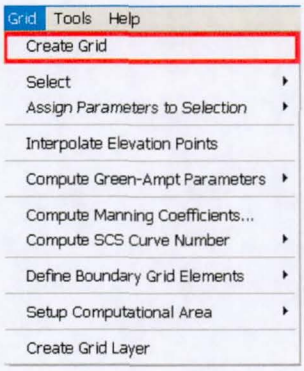
3.7 Start a new project from a *HEC-RAS* project file

If you have a *HEC-RAS* project file (*.prj) that includes channel reaches, you may use this option to import it to start a *FLO-2D* project. The working region and *FLO-2D* grid system are automatically scaled from the existing *HEC-RAS* project file extents.

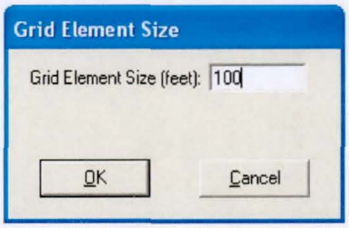


3.8 Grid System Generation

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



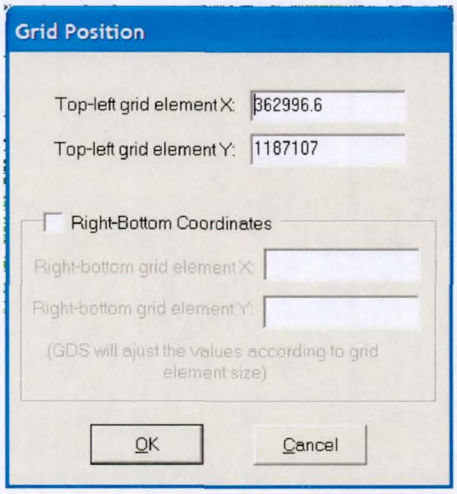
Input the appropriate grid element size (square grid) for the project.



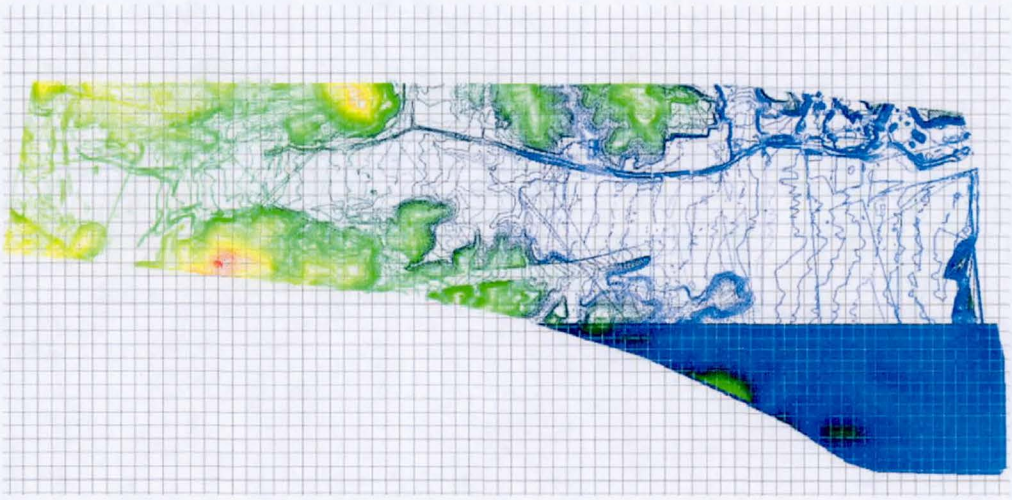
The selected grid element size should reflect the topographic data resolution and result in an acceptable number of total elements. A small grid element size enhances the *FLO-2D* model resolution but also increases the computer runtime and memory requirements. A balance should be sought between the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

IMPORTANT NOTE: ***A common mistake is to select a grid element that is too small! Smaller grid elements do not always significantly improve the model resolution, but can substantially increase the model simulation time. For large flood events where the entire floodplain is inundated by several feet or meters, the grid element size is not critical. The area of inundation depends more on the flood volume than on the grid element size. Using smaller grid elements help reducing the jaggedness of the flood boundary. Carefully review the guidelines for selecting a grid element size. A good rule of thumb is to select the grid element size such that the peak discharge divided by the surface area of one grid element is approximately less than one cfs/ft²:*** $Q_{peak}/A_{surf} < 1.0$ ***cfs/ft²***.

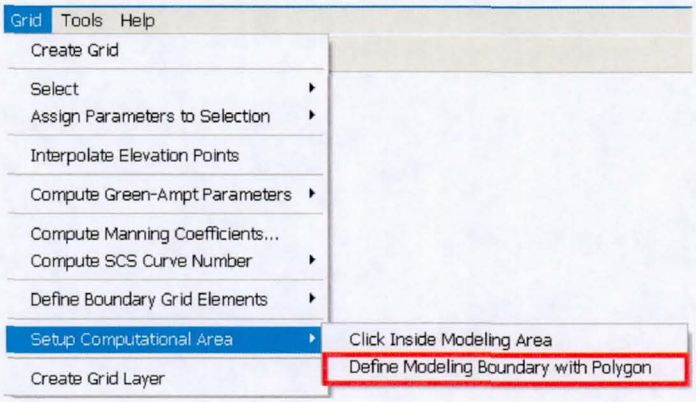
When you select the grid element size you will also have the option of changing the coordinates of the top left or/and bottom right grid element.



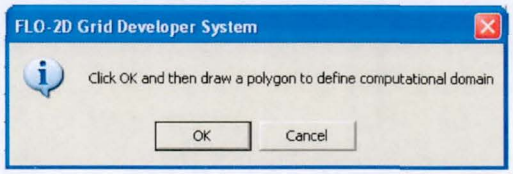
The following figure displays a system of 100 feet square grid elements overlaid on a set of DTM points that represent both random points and digitized contour lines.



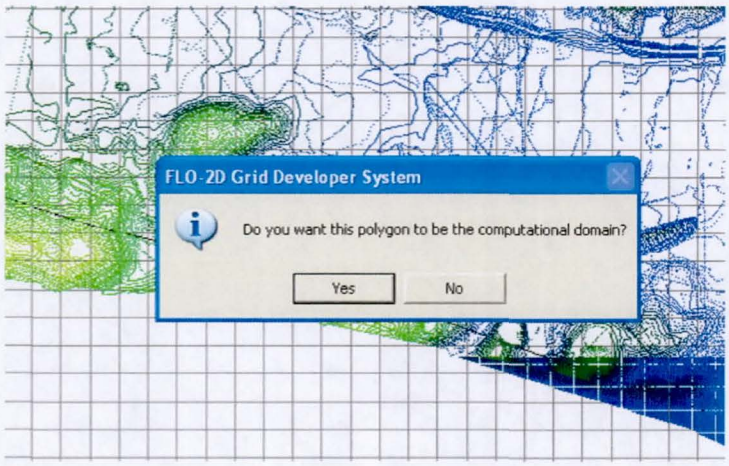
The computational domain can be defined by marking the boundary grid elements. The grid system can now outline your project area. The grid boundary is defined by clicking *Define Modeling Boundary with Polygon* option of the *Select Computational Area* in the *Grid* menu:



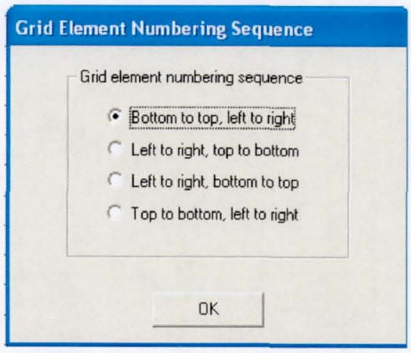
A dialog box appears instructing the user to draw the boundary polygon:



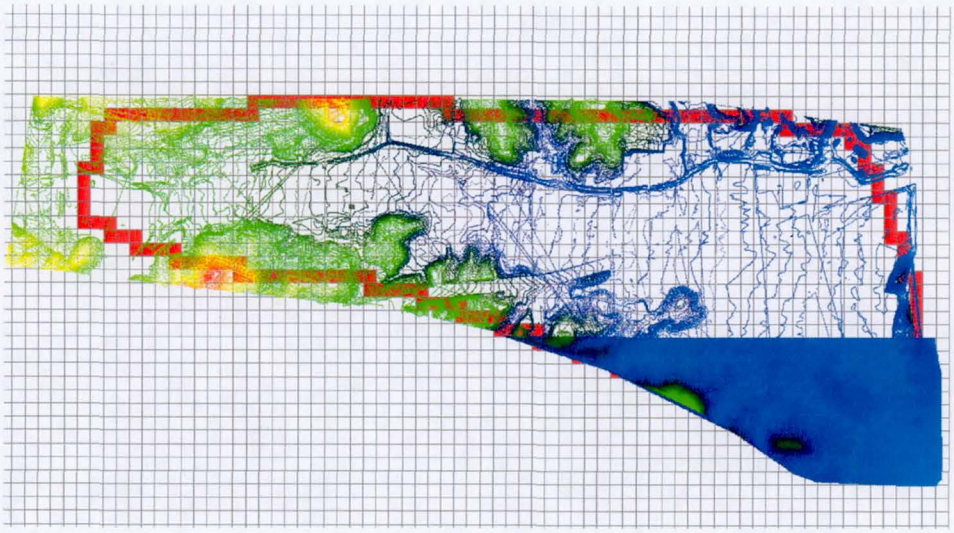
The polygon is completed by double clicking on the last polygon vertex before closing the polygon. When the user releases the mouse button, the system displays the message:



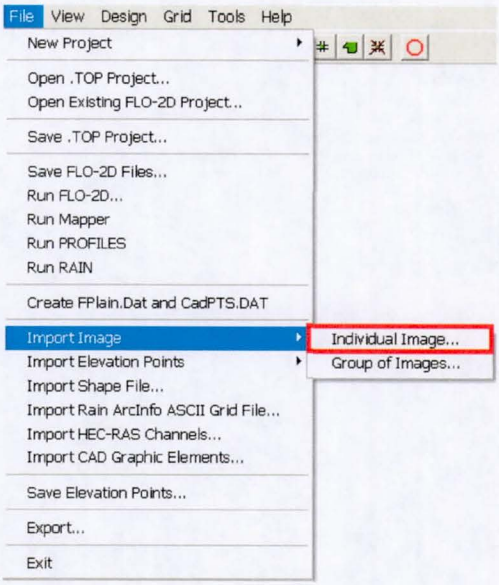
A 'No' response will erase the polygon while a 'Yes' response will proceed with creating the computation domain. A dialog box will appear so that the grid system numbering scheme can be selected:



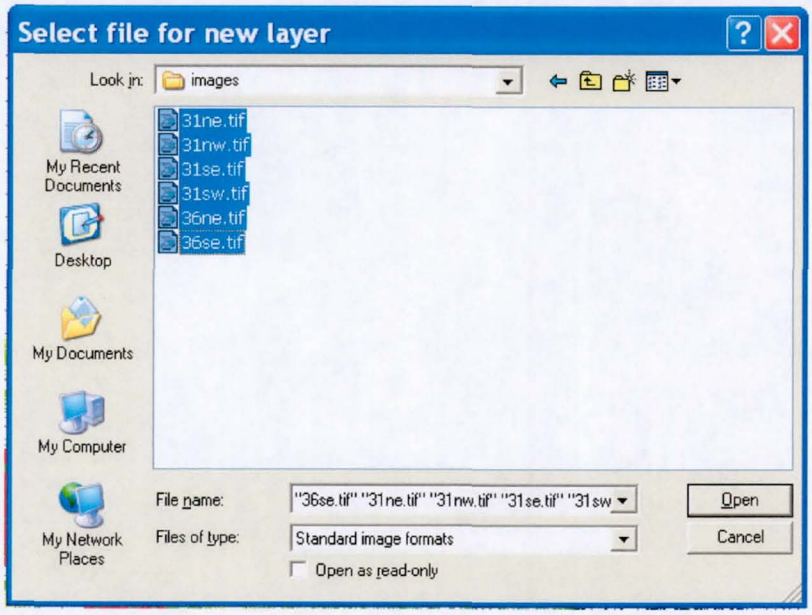
The numbering sequence is only important if you would need to match the grid with other grids to enlarge the computational domain. This may be necessary when working with very large projects (millions of grid elements). In that case a recommended numbering scheme is left to right, top to bottom if the drainage system runs from north to south or top to bottom of the screen. This enables the grid system to be expanded by more grid elements and maintain the number sequence. Click *'OK'* to generate the computational domain grid system:



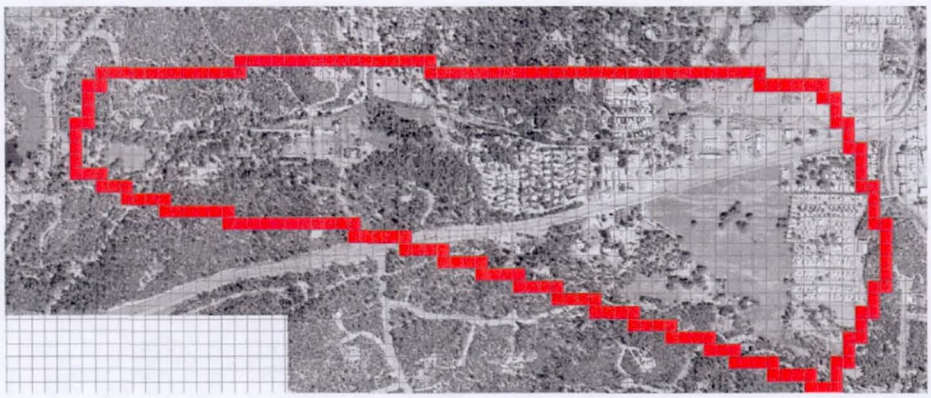
Importing a background image or having the DTM points displayed will enable you to visualize the computation domain and polygon boundary. To import a background aerial photo use *Import Image/Individual Image…* in the *File* menu:



Images can be selected one by one or several at a time using the Shift-click or Crtl-click on the files.



The following figure displays an example from a project with background aerial photos and the elevations points display turned off:



You may also define the computational boundary marking grid elements with by selecting the *Define Boundary Grid Elements Cells* command from the *Grid* menu or the icon. Click with the mouse on a grid element to mark a boundary element. The grid element will be colored red.

You may mark boundary elements one at time or by *painting* a line of elements by dragging the mouse while pressing the *shift* key. The grid boundary can be adjusted by adding or deleting them using the mouse. The boundary elements can be doubled-up or overlap. The objective is to have a completely enclosed flow domain. The boundary grid elements serve no other purpose than to define the *FLO-2D* computational domain. They have no function in a *FLO-2D* flood simulation, since only internal grid elements are used in the computations.

Upon completion, the computational domain will be the area inside the boundary grid elements. Please note that the computational domain is entirely enclosed by boundary grid elements creating a closed region. Click the *Select Computational Domain* command from the Grid menu and then use the mouse to click on any point inside the region. If this enclosure is incomplete (open), *GDS* will inform you that the area is still open and then additional boundary grid elements must be marked to close it.

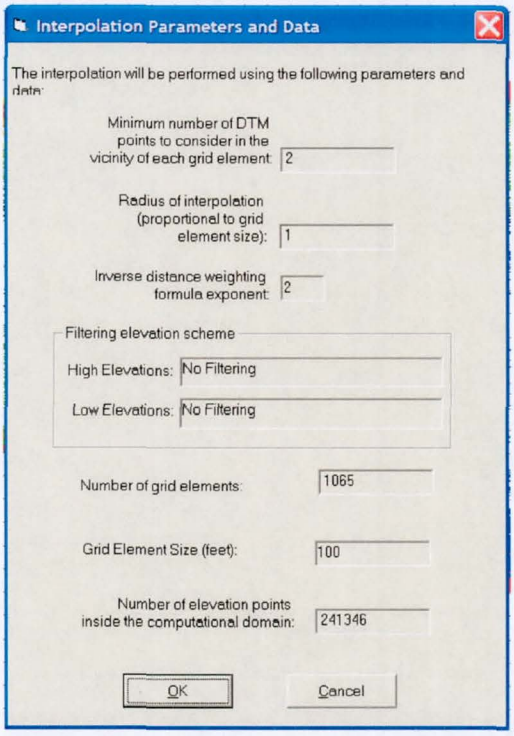
3.9 Assigning Manning's Roughness Coefficients (n-values)

You may input Manning's n-values to a group of grid elements. Start by clicking the *Select/Grid Element* command on the *Grid* menu and then click on the elements where you want to define the n-values. Selected elements will be hatched diagonally. Now choose the *Grid/Assign Parameters to Selection/Manning's Coefficients* command and input the desired n-value for the selected elements. By default all grid elements have an initial n-value assignment of 0.040. This default overland flow n-value is probably not representative of your project and you should spatially select and adjust the n-values for your project. It is possible to assign the n-values universally by selecting the *Inner Grid Elements* option under the *Select/Grid Element* command. The Manning's roughness values can also be computed from shape files; see the discussion under *Compute Manning Coefficients... Command (Grid Menu).*

3.10 Interpolation of Terrain Elevations

A set of DTM point elevations may be randomly distributed in the flow domain. However 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 *Option/Interpolation* command on the *Tools* menu.

The following table describes the various options for the interpolation algorithm:

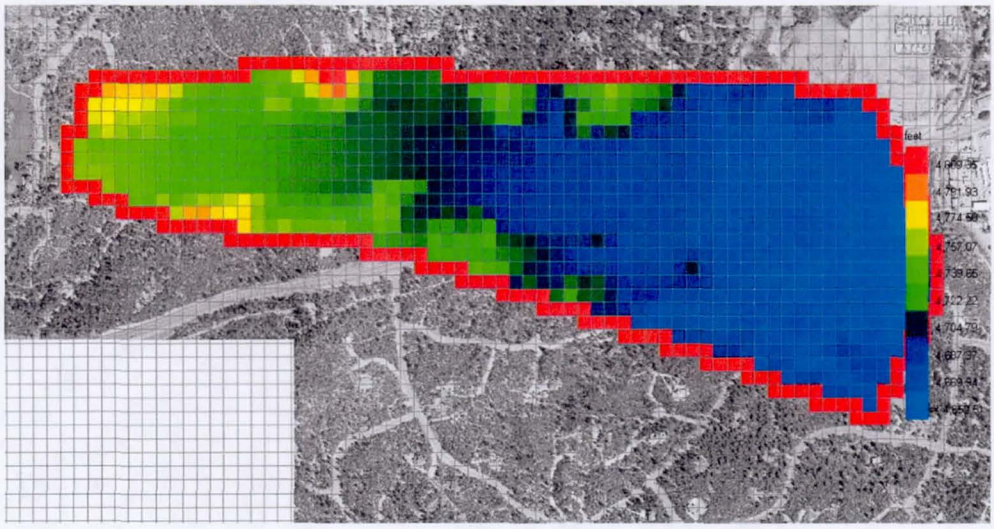
Option	Description
Minimum number of DTM points to consider in the vicinity of each grid element	To calculate the interpolated elevation that will represent each grid element, the algorithm analyzes at least the minimum number of points closest to the grid element node.
Radius of interpolation (proportional to grid element size)	This radius defines a circle around each grid element node based on a multiple of the grid elements side. Only DTM or assigned elevation points inside this radius are considered in the interpolation process. If no elevations point exists, the circle is automatically enlarged until at least the minimum number of points is found.
Inverse distance weighting formula exponent	This is "n" in the Inverse Distance Weighting Formula: $$\overline{Z} = \frac{\sum_{j=1}^{NDTM} Z_j \frac{1}{r_{ij}^n}}{\sum_{j=1}^{NDTM} \frac{1}{r_{ij}^n}}$$ Where $\overline{Z}$ is the interpolated grid element elevation, Z_j is the elevation of DTM point $_j$, r_{ij} is the distance from the DTM point $_j$ to center of grid element $_i$ and NDTM is the total number of DTM points.
No filtering (High or Low Elevations)	No filtering is performed on the elevation points.
Maximum elevation difference (High or Low Elevations)	For this option, the algorithm calculates a mean elevation using all the DTM and assigned elevation points within the interpolation radius. All those points that exceed the selected maximum elevation difference higher (or lower in the case of Low Elevation filter) than the mean are discarded and the mean elevation is recomputed and assigned to the grid element.
Standard deviation difference (High or Low Elevations)	The interpolation algorithm calculates the standard deviation of DTM point elevations and then neglects all those points whose elevations are higher (or lower in the case of the Low Elevation filter) than the standard deviation to determine the mean elevation to represent the grid element.

The high elevation filtering scheme has been designed to give greater weight to the floodplain elevations in computing a representative grid element elevation. Consider the case of an on-ramp to a highway that occupies only a portion of the grid system but happens to be located at the grid center. The grid element elevation would be based on the highway on-ramp elevation instead of the lower floodplain surface. This could result in an inappropriately high grid element elevation assignment.

By selecting the standard deviation option for the filter system, all the DTM points within the interpolation radius are used in the computation of the standard deviation. Then those DTM points that exceed the standard deviation elevation (higher than one standard deviation above the mean elevation) are discarded and the remaining points are averaged to compute the mean grid element elevation. It is also possible to filter out low elevation DTM points. Some DTM points may represent the channel bed elevation if the river is dry when the DTM data is collected. Grid element floodplain elevations along the river can be assigned elevations that are lower than desired by using the channel bed elevations. The lowest elevation DTM points (those on the channel bed) can be eliminated from the interpolation of the floodplain elevation by using the low elevation filter. These filtering schemes ensure that the ground surface is given preference when

assigning floodplain elevations. Once the grid element elevations have been assigned, double-click on a grid element to review its elevation and make any necessary changes.

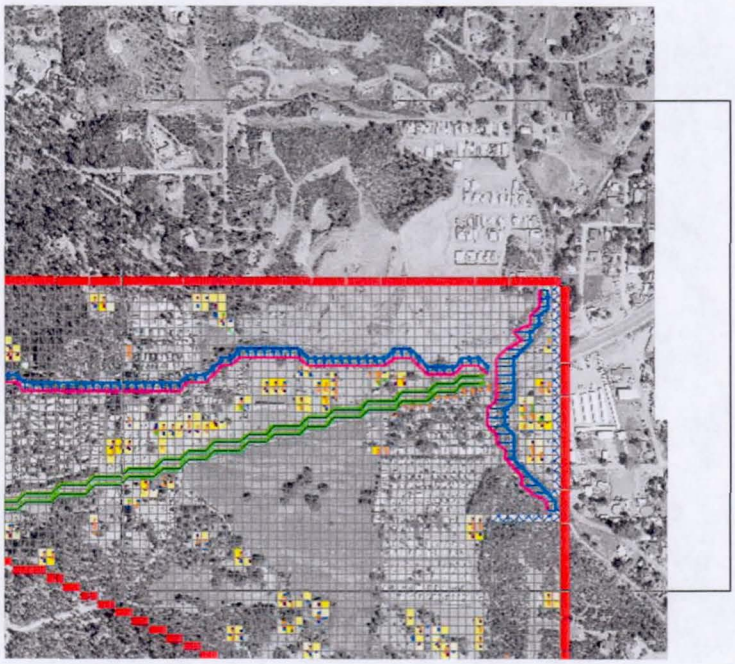
The following figures show the interpolated elevations represented using colors for each grid element. The red color represents the highest elevation while blue color the lowest elevation in the computational domain.



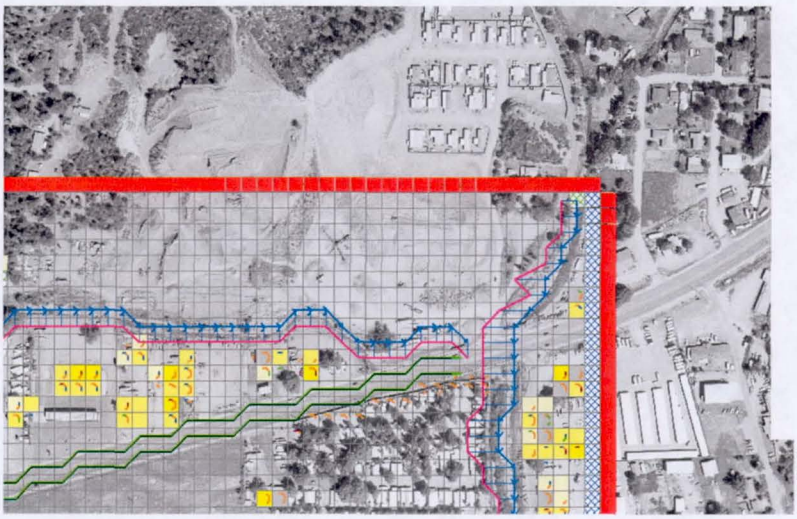
IV. Mouse Button Commands

4.1 Left Button actions (one-click)

You can zoom-in on the working region by selecting a rectangular area for a zoom view. Click the left mouse button to position the cursor on one the vertices of the desired area and drag the mouse to the opposite vertex. The mouse cursor changes to a cross-shaped pointer and a rectangle appears to outline the selected area.



When the button is released, the selected rectangular area is magnified to the full screen.



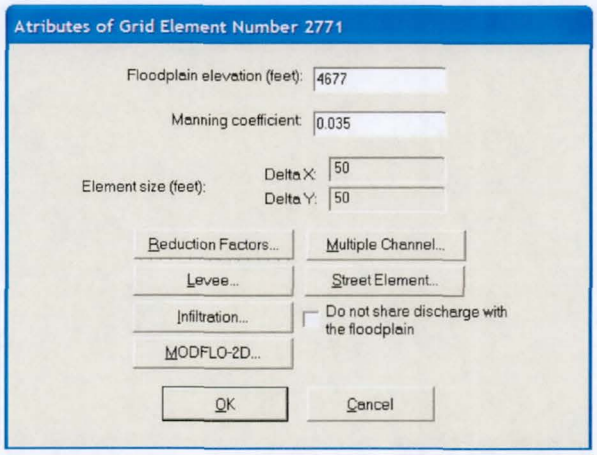
Clicking on a street will display a submenu that allows editing, modifying or deleting the street.

Clicking on a channel reach will display a submenu that also allows editing channel data.

How to edit streets and channel data is explained in chapter 5.

4.2 Left Button actions (double-click)

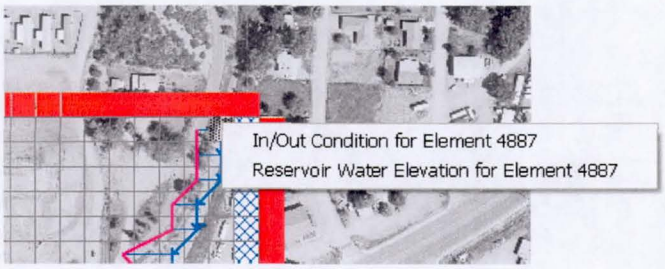
After the grid system has been created and the grid element elevations assigned, you can double-click on a grid element to display the following dialog box:



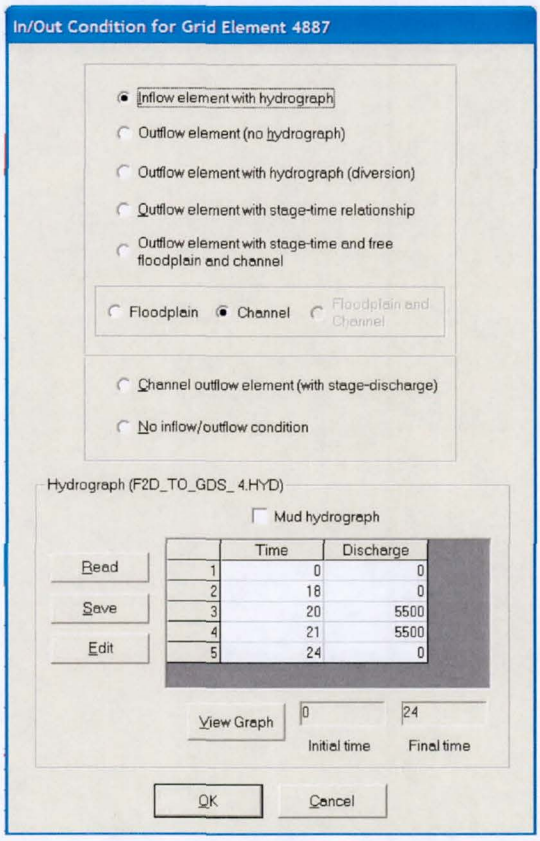
You can enter a new elevation or n-value for any grid element. There are buttons to edit the grid element attributes such as levees, infiltration, streets, etc.

4.3 Right Button actions

After the grid system has been created and the grid element elevations assigned, you can right-click on a grid element to display the following short cut menu:



Clicking on this short-cut message, the inflow/outflow dialog box is displayed:



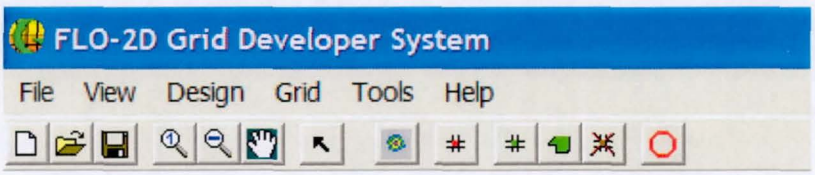
The input description for the FLO-2D inflow or outflow data is explained in chapter 5.

V. *Toolbar and Menus*

This section describes the *GDS* commands in the information toolbar and main Menu.

5.1 Toolbar

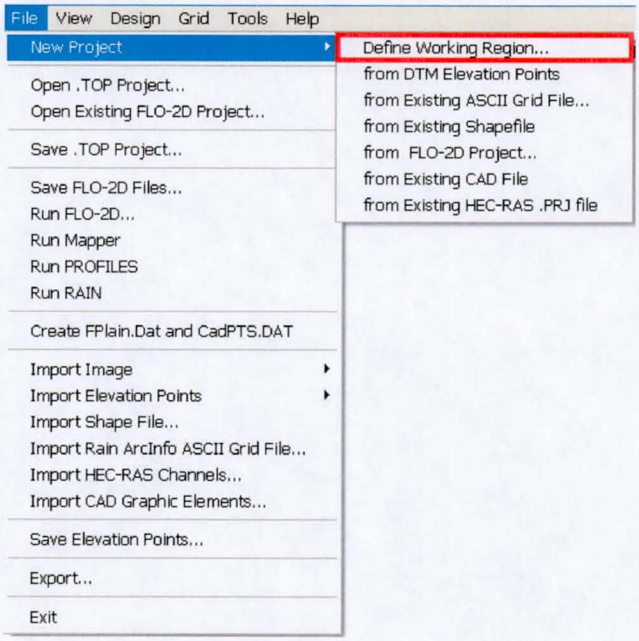
Toolbar Icon	Function
	Start new project
	Open existing project (.TOP file)
	Save project (.TOP file)
	View extent of modeling region (100% view)
	Zoom out to previous view
	Pan view
	Select
	Insert elevation point
	Define boundary grid element by grid element
	Select grid element by grid element
	Select grid elements defined by polygon
	Unselect all grid elements
	Levee express editor



5.2 File Menu Commands:

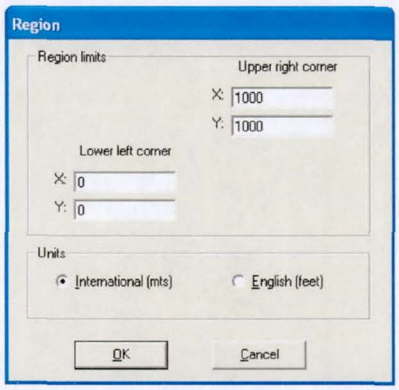
5.2.1 Define Working Region Command (File Menu)

This command creates a new project.



You will then be prompted for the coordinates that define the project working region.

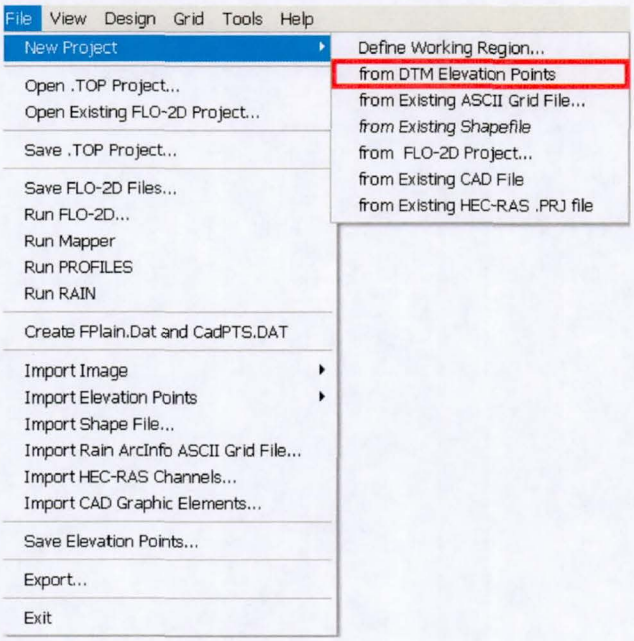
IMPORTANT NOTE: You can only have one project open at a time. If there is a project open when this command is selected, the GDS will prompt you to save the current project.



The following table explains the required variables.

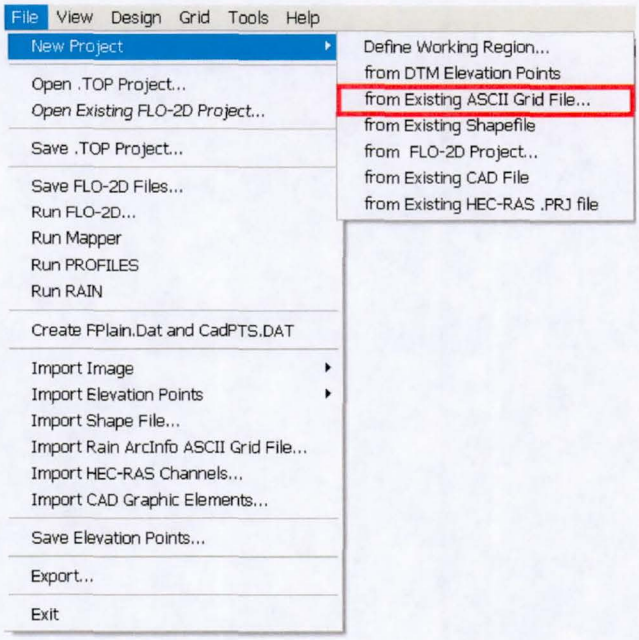
OPTION	DESCRIPTION
X (Upper Right Corner)	The X coordinate of the upper right hand corner that defines the working region.
Y (Upper Right Corner)	The Y coordinate of the upper right hand corner that defines the working region.
X (Lower Left Corner)	The X coordinate of the lower left hand corner that defines the working region.
Y (Lower Left Corner)	The Y coordinate of the lower left hand corner that defines the working region.

5.2.2 New Project/from DTM Elevation Points Command (File Menu)



A new project is created by importing the DTM elevation points from an existing file. To import the DTM (*.PTS) file click on this command in the *File* menu and chose the correct filename in your project subdirectory. The working region is automatically scaled from the minimum and maximum point coordinates.

5.2.3 New Project/from Existing ArcInfo ASCII Grid File... Command (File Menu)



Using this command, a new project is created by importing the terrain elevation points from an existing ArcInfo ASCII grid file. The format of these files is as follows:

```
ncols /* Number of columns in the grid */
nrows /* Number of rows in the grid */
xllcorner x /* Lower left x coordinate of grid */
yllcorner y /* Lower left x coordinate of grid */
cellsize size /* Grid cell size */
NODATRA_value NODATA /* value of an empty grid cell */
z11 z12 z13 ... z1ncols /* values of row 1 */
z21 z22 z23 ... z2ncols /* values of row 2 */
...
...
znrows1 znrows2 znrows3 ... znrowsncols /* values of last row*/
```

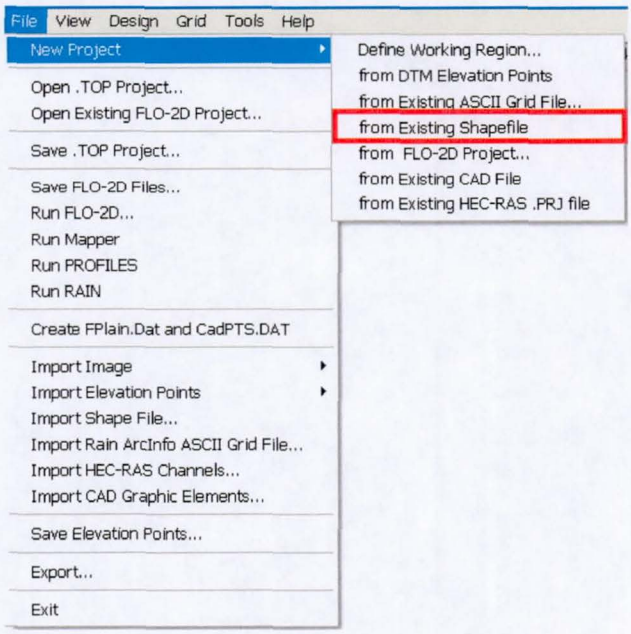
Rows are read from north to south. For example:

```
ncols        388
nrows        461
xllcorner    674070.85270015
yllcorner    1000118.1562353
cellsize     100
NODATA_value -9999
```

2477.259 2480.868 2486.877 2486.877 2487.308 2490.641 2493.438 2493.438 2493.438. . .

The project area is automatically scaled from the minimum and maximum point coordinates. To import the ArcInfo ASCII Grid File (*.ASC) file click on this command in the *File* menu and choose the correct filename in your project subdirectory.

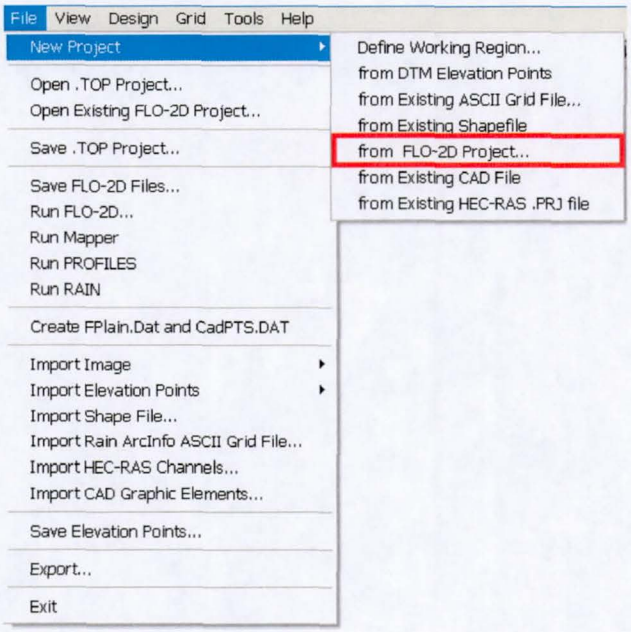
5.2.4 New Project/from Existing Shapefile…Command (File Menu)



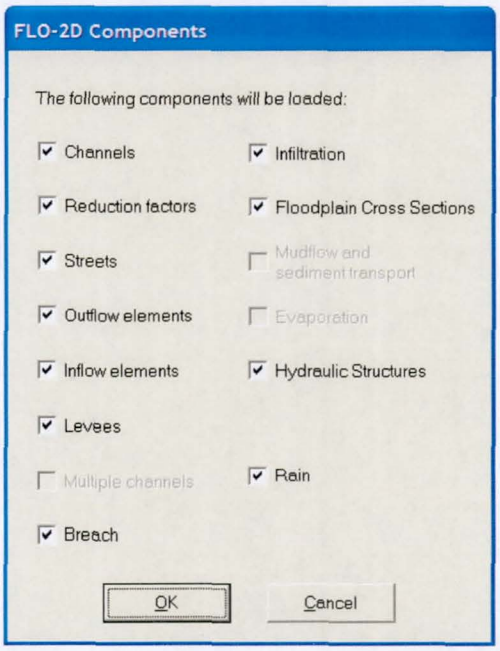
A new project can be created by importing an ESRI *PointZ* Shape file. The working region is automatically scaled from coordinate data in the shape file. You can only have one project open at a time. If there is a project open when you select this command, the *GDS* prompts you to save the current project.

IMPORTANT NOTE: GDS is only able to extract data from POINTZ shape files, not from polygon or line shape files. .

5.2.5 New Project/from *FLO-2D* Project…Command (File Menu)

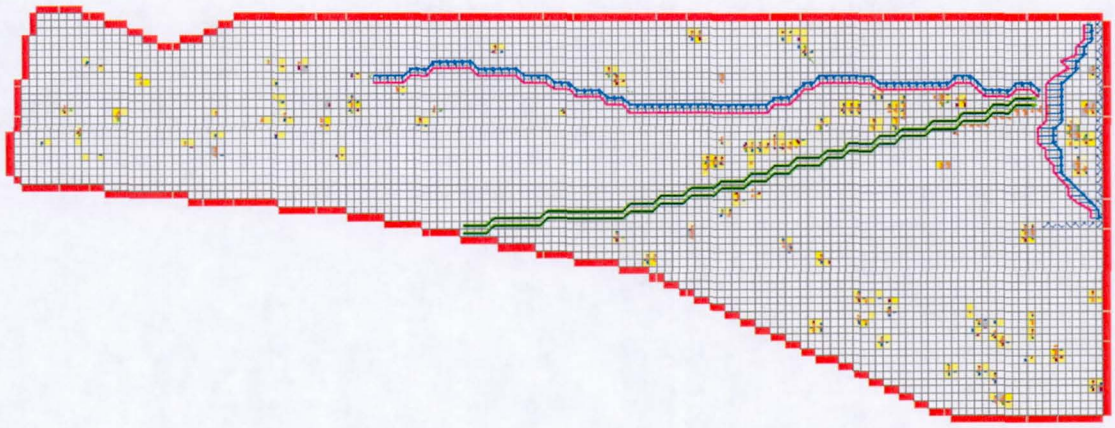


A new project grid system can be created by importing an existing *FLO-2D* project. The working region and the *FLO-2D* grid are automatically scaled from the minimum and maximum coordinate points. To use this option, click on this command in the *File* menu and chose the FPLAIN.DAT file in your project subdirectory. The CONT.DAT data file must exist in the subdirectory. An existing project may have various components that have already been developed such as channels, streets, levees, etc. The components that you want to import can be identified in the following dialog box:

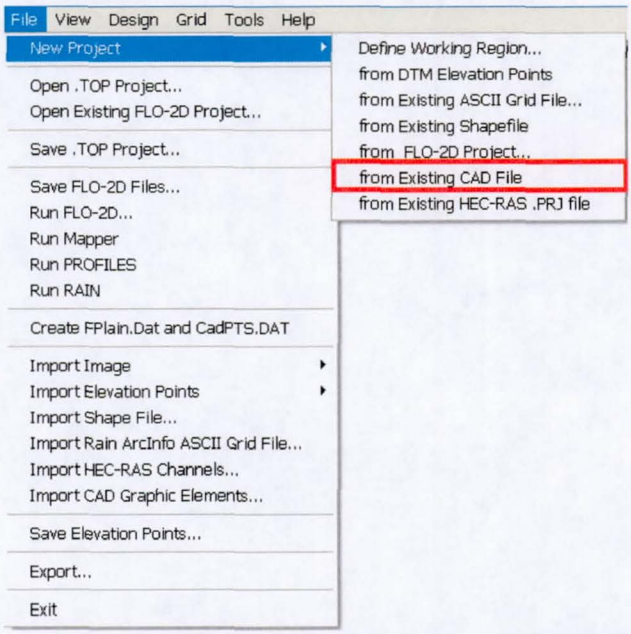


Only available components will be available, otherwise the corresponding check boxes will be unavailable.

Existing component data (such as reduction factors or levees) can be graphically edited by simply clicking on the grid element with the left mouse button. A dialog box will appear so that you can select the component for editing. This editing procedure is different from the procedure creating new components such as streets or channels. The resulting grid may look like this:



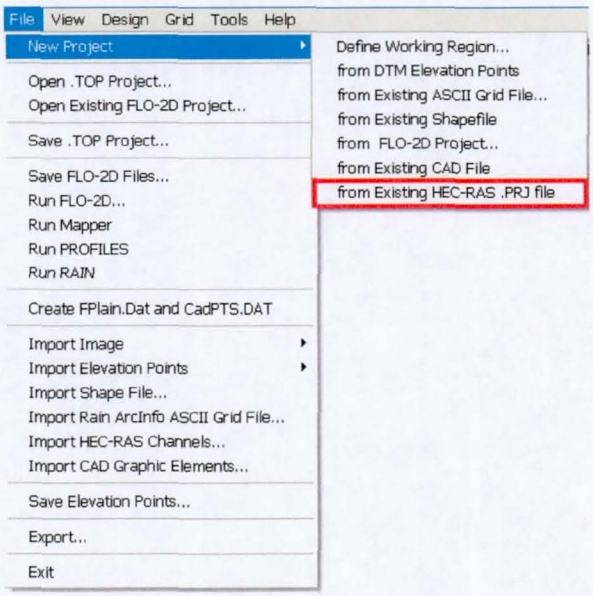
5.2.6 New Project/from Existing CAD File Command (File Menu)



This option provides a way to start a *GDS* project by importing a DXF or DWG CAD files. The working region and *FLO-2D* grid system are automatically scaled from the existing CAD file extents.

5.2.7 New Project/from Existing *HEC-RAS* .PRJ File Command (File Menu)

You may use this option to import it to start a *FLO-2D* project when you have a *HEC-RAS* project file (*.prj) that includes channel reaches. The working region and *FLO-2D* grid system are automatically scaled from the existing HEC-*RAS* project file extents.



5.2.8 Open .TOP Project… Command (File Menu)

Use the *Open .TOP Project* command to open an existing project. A dialog box will help you find the directory and the project file. All *GDS* project files have a *.TOP extension.

5.2.9 Open Existing *FLO-2D* Project … Command (File Menu)

This command is equivalent to the *New Project/from FLO-2D Project* command in the File menu.

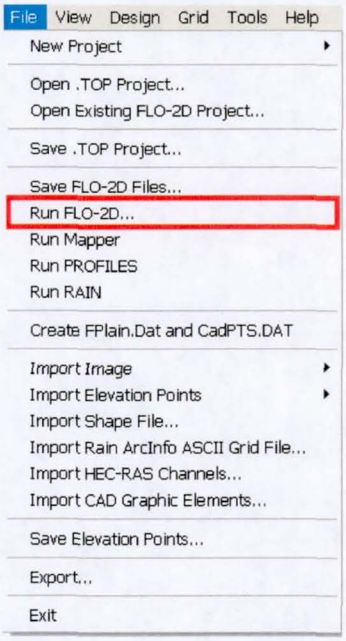
5.2.10 Save .TOP Project… Command (File Menu)

Use this command to save the active project. The system will display a file dialog so that a directory and file name can be selected. The project file name must have a *.TOP extension.

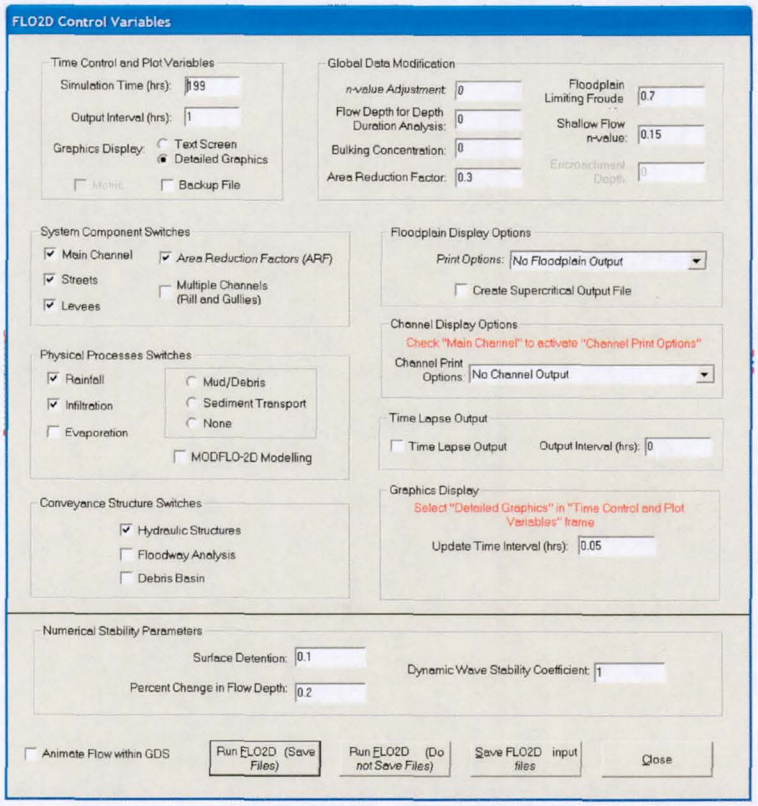
5.2.11 Save *FLO-2D* Files Command (File Menu)

This command allows you to save the *FLO-2D* data files for use with the *FLO-2D* model.

5.2.12 Run *FLO-2D*... Command (File Menu)



This command allows you to run the *FLO-2D* model from the *GDS*. When you click on Run *FLO-2D* the following dialog box will be displayed:



In this dialog box, the control parameters can be selected for a *FLO-2D* simulation. To start a basic overland flood simulation, you only need to input the project simulation time (SIMUL) and the output interval (TOUT) in the Time Control and Plot Variables frame. Also check *Detailed Graphics* if you want *FLO-2D* to display flood graphics during the model run. To start the model, click on the *Run FLO2D* button. To save the *FLO-2D* input files (.DAT files) without running the model click on the *Save FLO-2D input files* button.

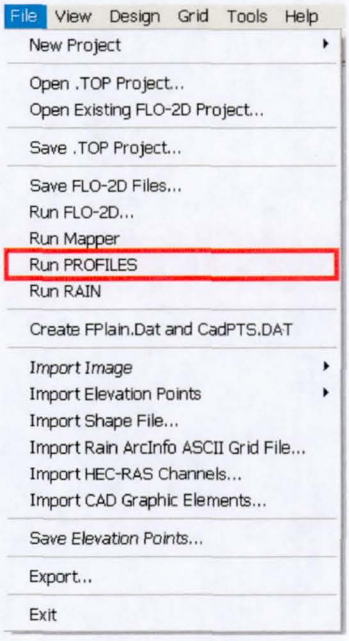
Check the *Animate Flow within GDS* to plot animated flow depth while the model is running. This feature displays the animated flooding over background aerial photos. The *FLO-2D* simulation may be slowed down due to the graphic display of the aerial photo. If only simple animation is required without background images, unselect the check button (as shown above) and the model simulation will run faster.

5.2.13 Run Mapper Command (File Menu)

This command initiates the *Mapper* post-processor program to create flood hazard maps. Detailed instructions of the *Mapper* program are presented in the separate Mapper manual.

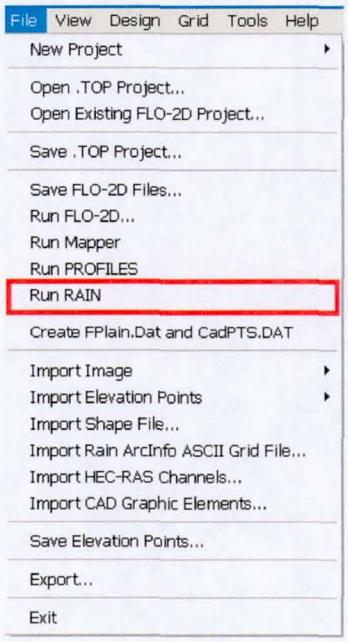
5.2.14 Run PROFILES Command (File Menu)

Use this command to run the PROFILES program, used to interpolate channel cross sections and slopes.



5.2.15 Run RAIN Command (File Menu)

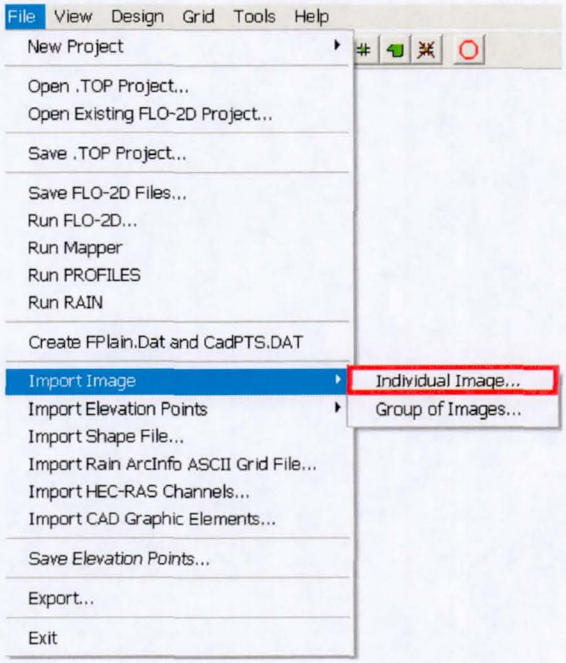
Use this command to run the RAIN program. This option is not activated for this *FLO-2D* version.



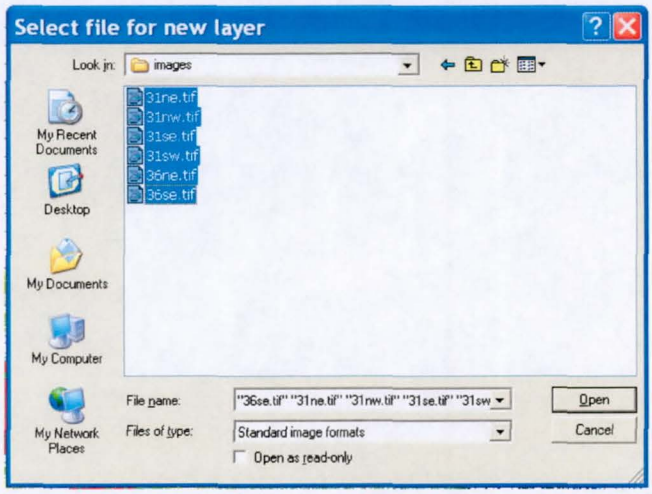
5.2.16 Create FPLAIN.DAT and CADPTS.DAT Files Command (File Menu)

This is an optional command to create only FPLAIN.DAT and CADPTS.DAT files required by *FLO-2D*. These two topographic files are then created separately without the other required *FLO-2D* files. This is procedure is appropriate when creating a grid system in segments for later compilation.

5.2.17 Import Image/Individual Image Command (File Menu)



Use this command to import individual images such as aerial photos. You may select images one by one or multiple images at a time using the Shift-click or Crtl-click on the files.



You may import images that have been created in following formats:

File Type	Description	Common Extensions
ARC/INFO Grid	ArcInfo GRID files	*asc, *.prj
ADRG	Digitized Raster Graphic	*.img, *.ovr,*.arc
ASRP/USRP	DIGEST ASRP, A NATO Military format	*.img, *.ovr, *.

BIL	Band interleaved by line multiband images	*.bil
BIP	Band interleaved by pixel multiband images	*.bip
BMP	Windows bitmap	*.bmp, *.dib
BSQ	Band sequential multiband images	*.bsq
CADRG	Compressed Arc Digitized Raster Graphics	*.*
CIB	Controlled Image Base	*.*
CRP	Compressed Raster Product (Military GeoTIFF)	*.tif
ERDAS/IMAGINE		*.gis, *.lan
GeoTIFF	TIFF with a Geo header	*.tif, *.tff, *.tiff
GIF	Graphics Interchange Format	*.gif
Image Catalogs	Image catalog (collection of images)	*.*
IMPELL RLC	Run-length compressed files	*.rlc
JPEG	JPEG	*.jpg, *.jpeg
MrSID	Multi-Resolution Seamless Image Database	*.sid
NITF	National Imagery Transfer Format	*.ntf
Sun raster file		*.rs, *.ras; *.sun
SVF	Single Variable File	*.svf
TIFF	Tagged Image File Format	*.tif, *.tff, *.tiff

To correctly locate the image or photo, it must be accompanied by a world file that contains geo-reference data. This world file has an extension depending on the image and file type and according to the table below. For example, if you have an image with a file name *myimage.bmp*, then the world file associated with it must be named *myimage.bmpw* or *myimage.bpw*

File Extension	World File Extension
bmp	bmpw or bpw
jpg; jpeg	jpgw or jgw
tif; tff; tiff	Tfw
gis	Gsw
lan	Lnw
bil	Blw
bip	Bpw
bsq	Bqw
sid	Sdw
sun	Snw
rs; ras	Rsw
rlc	Rcw

The world file has the following general format:

Line 1: This line has the dimension of a pixel in map units in the x-direction.
Lines 2 and 3: These lines are the rotation terms (Not used in this release).
Line 4: This value is always negative because the image space is top-down whereas the map space is bottom-up.

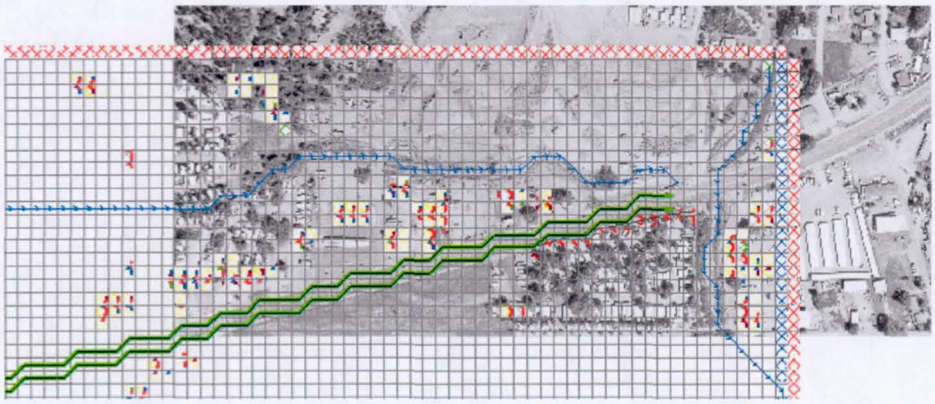
Line 5: This line has the translation term; x-Origin (x-coordinate of the center of the upper left pixel).

Line 6: This line has the translation term; y-Origin (y-coordinate of the center of the upper left pixel).

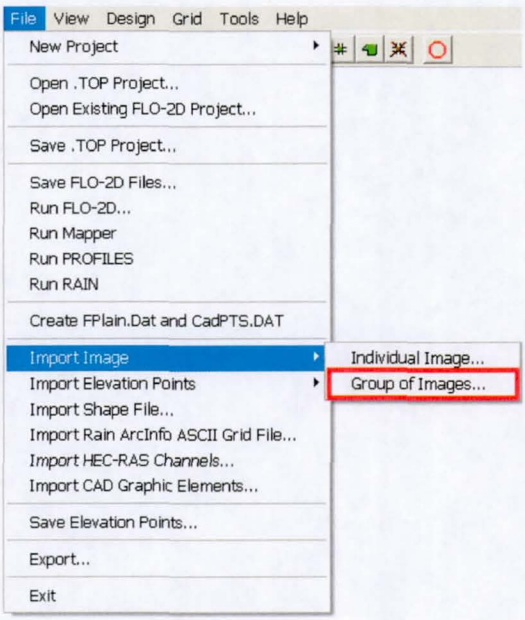
An example world file format is:

20
0
0
-20
637510
1032490

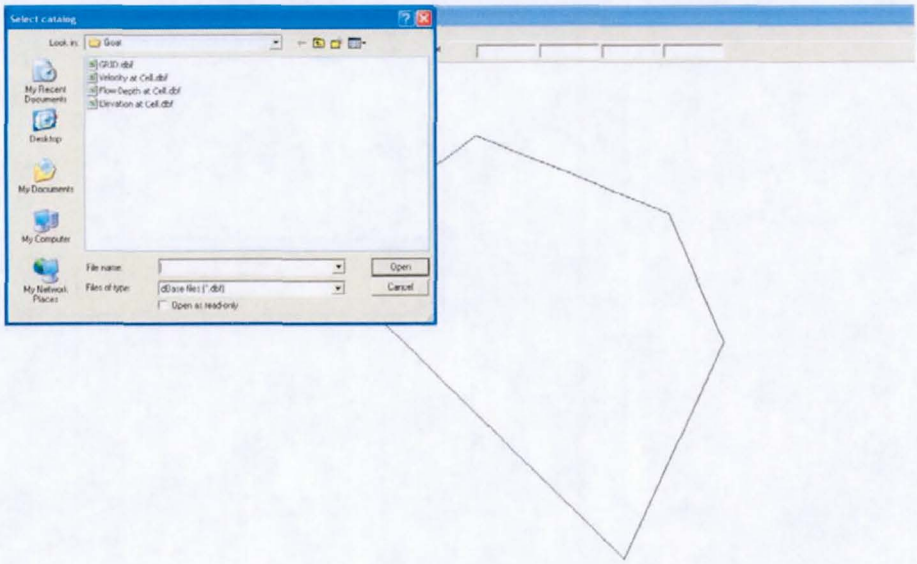
An imported image is illustrated in the following figure:



5.2.18 Import Image/Group of Images Command (File Menu)



This command allows importing of several image files contained in a given subdirectory. First draw a polygon on the working region and then select an image catalog file.

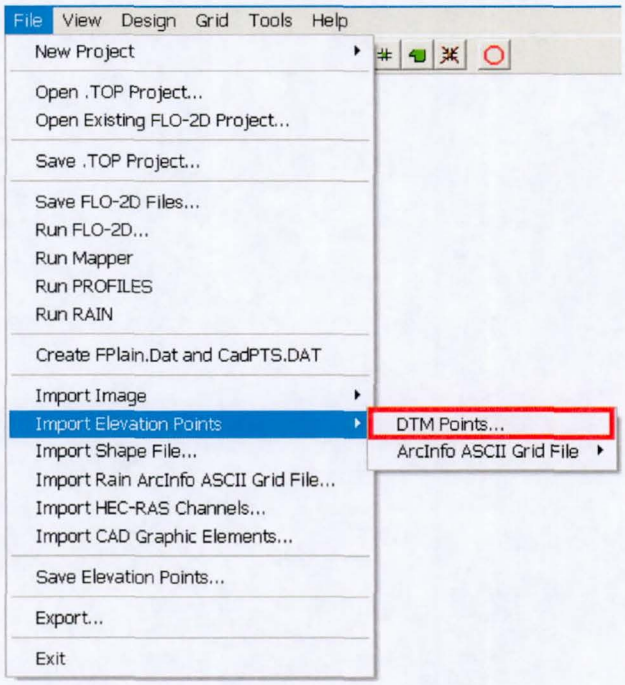


The catalog file may be in DBASE or ASCII format and has the following format:

Image	Xmin	Ymin	Xmax	Ymax
C:\Projects\MaricopaCounty\Data\ 6401030-5.TIF	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\ 6401035-5.TIF	660000	1300000	770000	1500000
\\Agua\IMF\Publico IMF\ 6401055-5.TIF	630000	1300000	750000	1510000

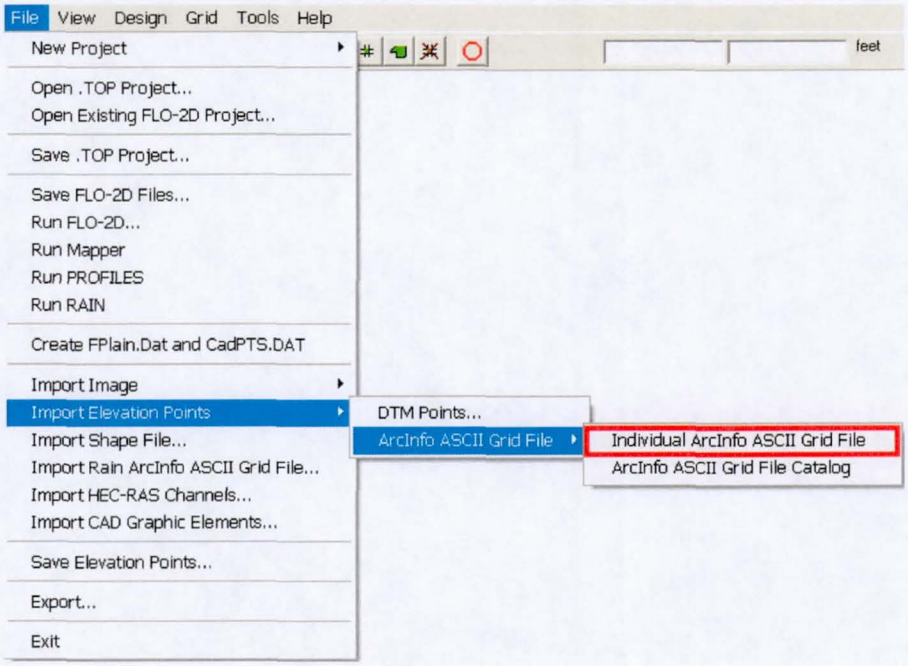
The first column is the file name including its path and the following four columns are the image coordinate limits. The *GDS* will find all images from the catalog that are contained within or intersected by the user defined polygon and will retrieve the corresponding images.

5.2.19 Import Elevation Points/ DTM Points… Command (File Menu)



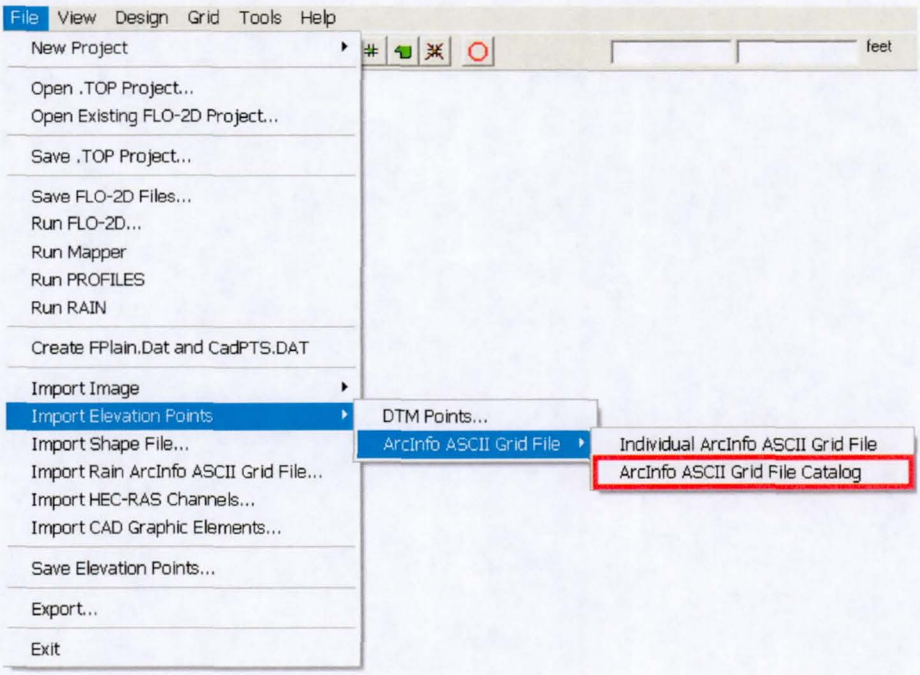
This command imports DTM elevation points from an existing file. You may import several data files and the new points are appended to the existing data points. You may also mix or combine DTM points with points from ArcInfo ASCII grid files.

5.2.20 Individual ArcInfo ASCII Grid File Command (File Menu)



Use this command to import ArcInfo ASCII grid files. You may import several grid files. Any new points are added to the existing data. You may also mix or combine ArcInfo ASCII data with DTM points.

5.2.21 ArcInfo ASCII Grid File Catalog Command (File Menu)

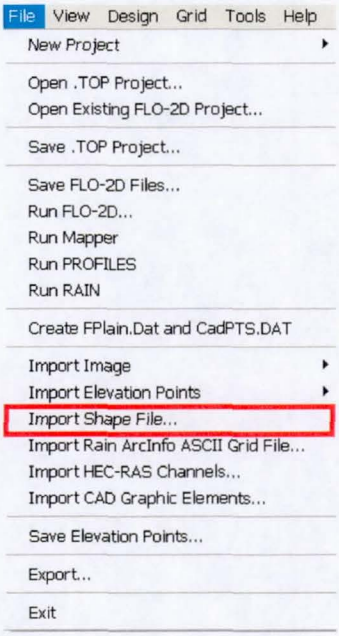


With this command you can import several ArcInfo ASCII grid files stored in any subdirectory. First draw a polygon on the working region, then select a catalog file. The catalog file may be in DBASE or ASCII format and has the following format:

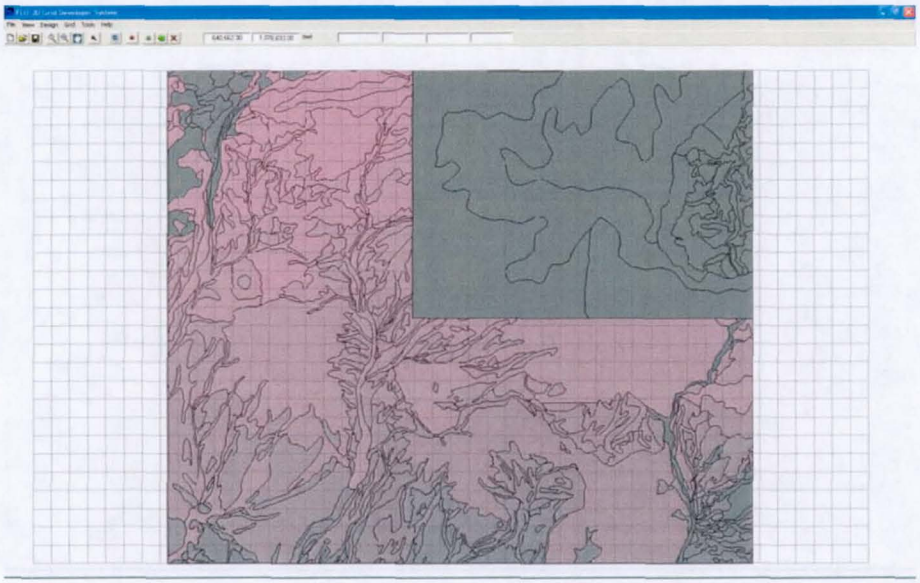
ASCII Grid File	Xmin	Ymin	Xmax	Ymax
C:\Projects\MaricopaCounty\Data\grd2375-100.asc	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\grd2376-100.asc	500000	1000000	900000	1500000
\\Agua\IMF\Publico IMF\grd2476-100.asc	510000	1000000	740000	1510000

The first column is the file name including the path name. The following four columns are the coordinate data limits in each file. The *GDS* will automatically find all data files that are contained within or intersected by the user defined polygon and will retrieve/display the corresponding files. Grid files generally contain a large number of points and loading may take several minutes, but once loaded, the points are quickly displayed. If the display time becomes too long for a given large data point set, you can use the *Esc* key to stop the display process.

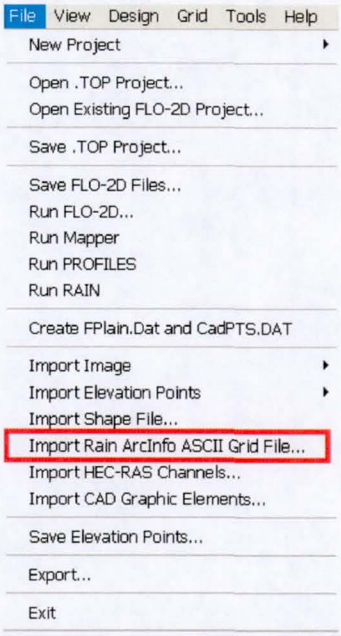
5.2.22 Import Shape File Command (File Menu)



This command will import ESRI shape files. The shape file can represent soil properties, land use, Manning's n-values or other variables. The shape file below displays n-values.



5.2.23 Import ArcInfo ASCII Grid File (File Menu)



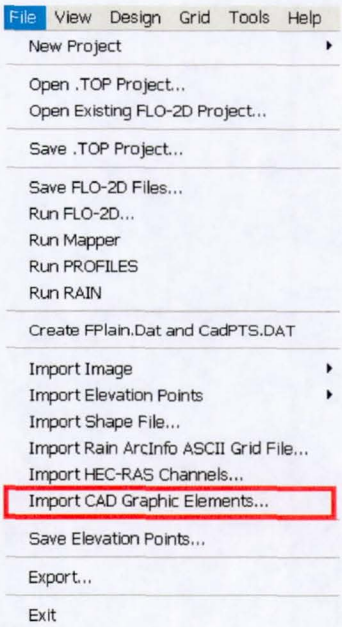
An ASCII grid file can be imported as a *FLO-2D* grid system or used to delineate other boundaries such as a rainfall gage grid system.

5.2.24 Import *HEC-RAS* Channels… (File Menu)

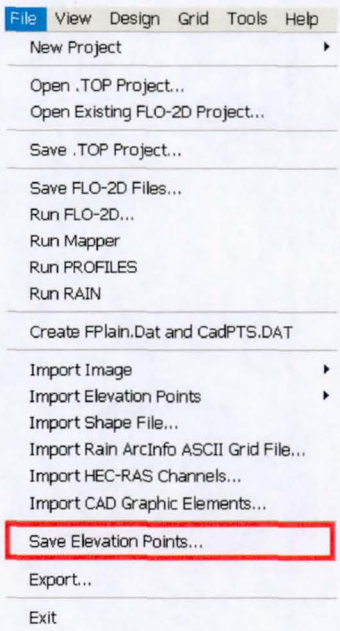
This command allows importing channel reaches from *HEC-RAS* geometric files.

5.2.25 Import CAD Graphic Elements… (File Menu)

Use this command to import DXF or DWG CAD files.

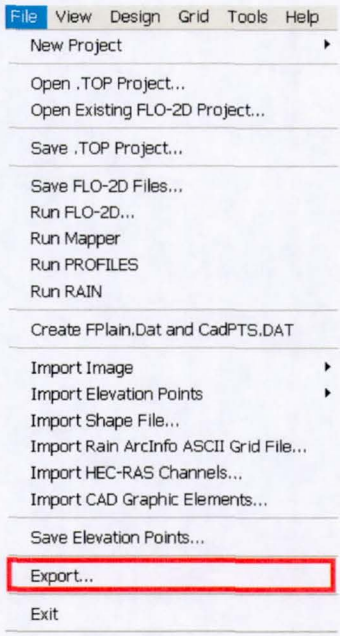


5.2.26 Save Elevation Points (File Menu)

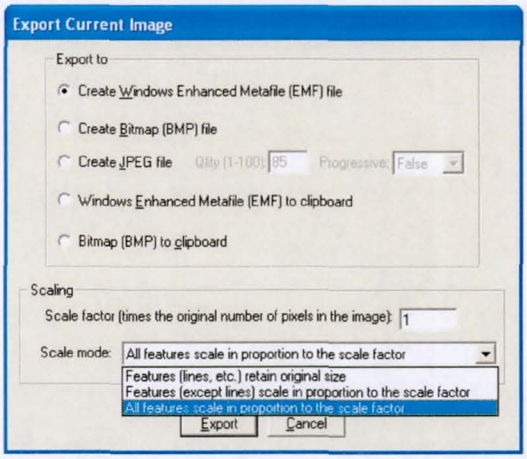


After deleting any DTM points, this command will save the remaining points to a file.

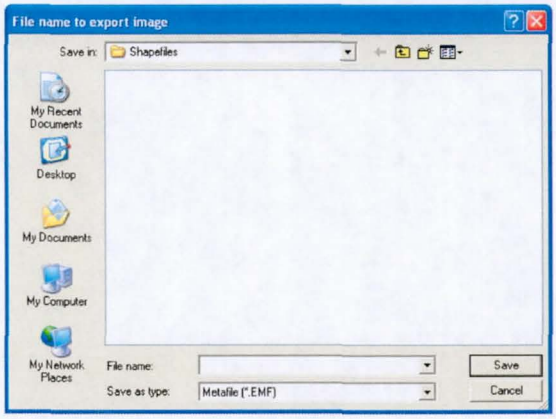
5.2.27 Export…Command (File Menu)



Use this command to export the current screen view to one of various image formats. Click on the *Export Command* and the following dialog box appears:



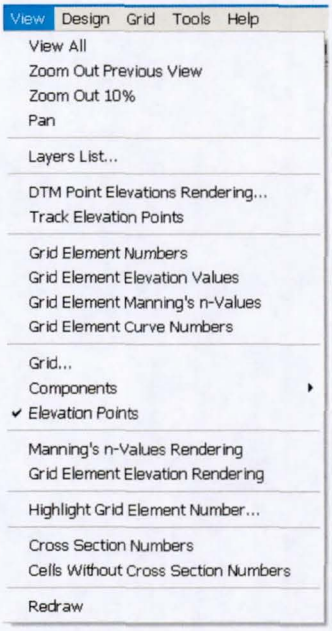
Choose an image export format such as Bitmap, JPEG, etc and adjust the scale factor of the image to obtain better print quality. When you click on the Export button, input the image file name and select the directory to save the image as shown in the following file dialog box:



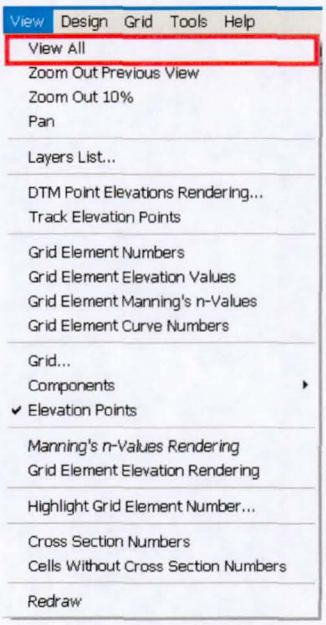
5.2.28 ***Exit*** Command (File Menu)

This command ends your *GDS* session. *GDS* will prompt you to save the current project.

5.3 *View* Menu Commands:



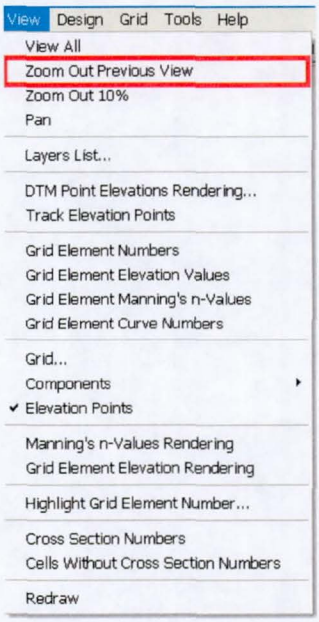
5.3.1 View All Command (View Menu)



Use the *View All command* or click on the View All icon to return the working region to its original full size. To Zoom-in (increase the magnification), just

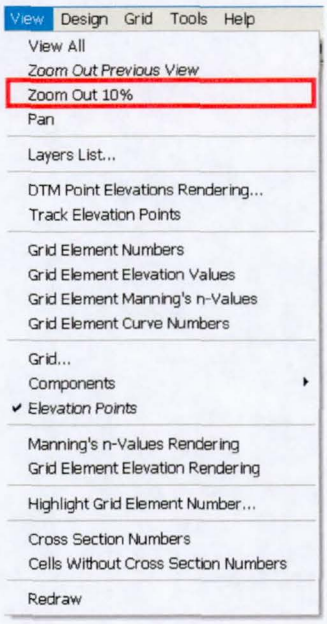
click in the working region and drag the mouse to outline the area you want to magnify.

5.3.2 Zoom Out Previous View Command (View Menu)



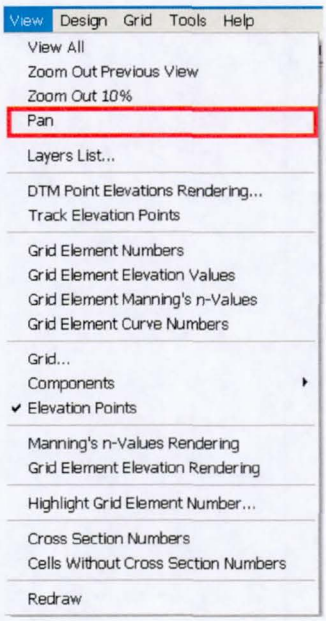
Use the *Zoom Out Previous View* command to return to the previous zoom extent

5.3.3 Zoom Out 10% Command (View Menu)



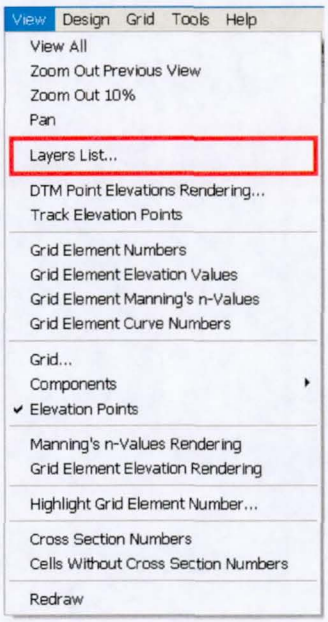
Use the Zoom Out 10% View command to reduce the current view 10% in size.

5.3.4 Pan Command (View Menu)

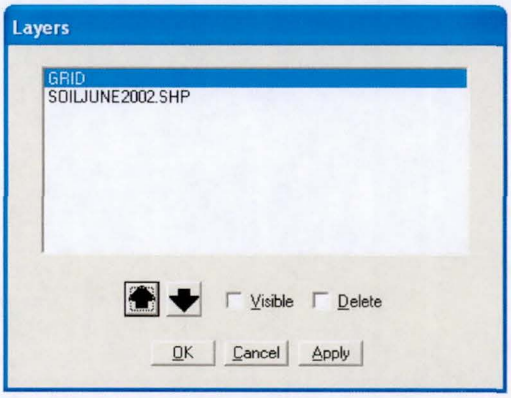


Use the *Pan* command to move around within the working region view. Click and drag the mouse to pan around. You can also use the *Pan* Toolbar icon for the same purpose. Use the *View All* command (or toolbar icon) to return to a full view of the working region.

5.3.5 Layers List Command (View Menu)

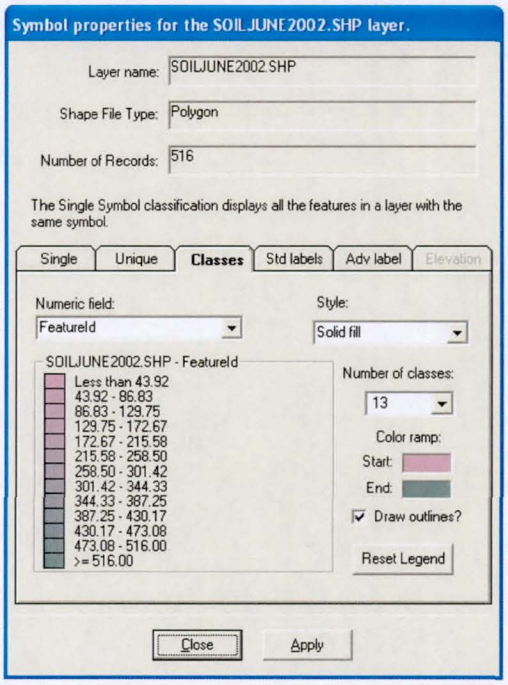


This command opens the Layer dialog box.



Layers may be visible or invisible. You can change the layer visible status by checking the *Visible* check box. In the example above there are two active layers. You may use the and icons to highlight and move between the various layers. Click on the *Apply* button to accept the changes. You may also delete any layer by checking the *Delete* check box and then clicking the *Apply* button.

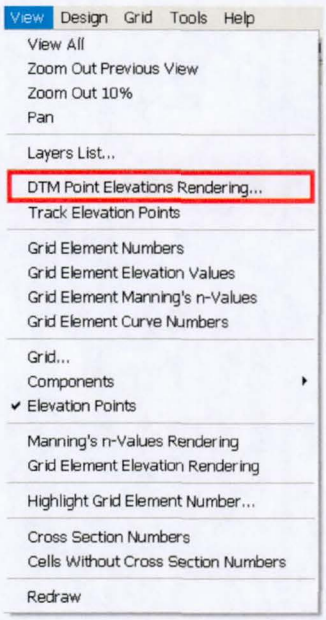
To enter the Layer Management Dialog box double click on an active layer. The following dialog appears:



In this dialog box you may change layer colors, transparency, number of classes or divisions, add point captions, font styles, etc. Using the *Adv label* option you may add labels to DTM points in the Elevation layer.

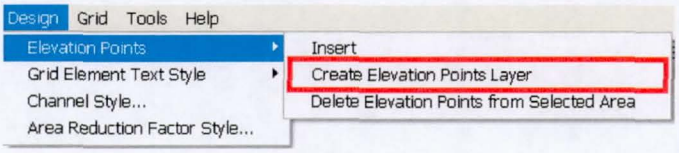
Click on the *Apply* button and then the *Close* button to accept the changes.

5.3.6 DTM Point Elevations Rendering Command (View Menu)

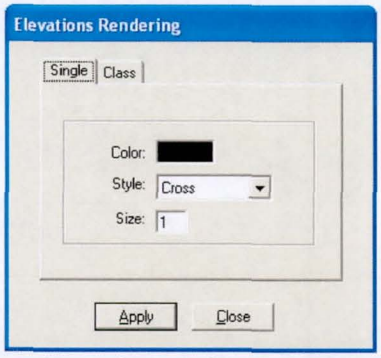


To optimize display times the *GDS* does not automatically create the Elevation layer. For certain functions, however, like the *Elevation Rendering* this layer is required. If the layer does not exist, you will get the following message:

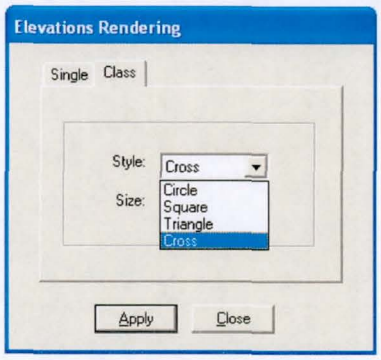
To create the layer, use the *Create Elevation Point Layer* command in the *Design* menu:



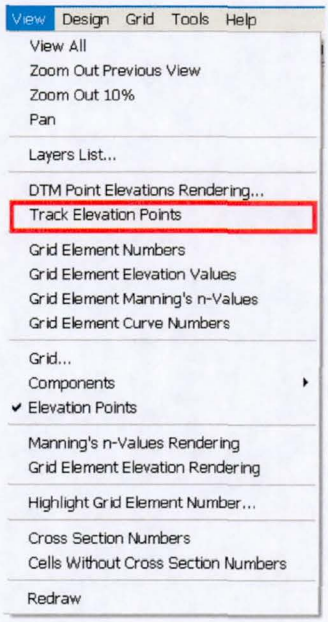
After the elevation layer is created, click on Elevation Rendering again to get the dialog:



The "Single" tab selects a single color, style, and size for all elevations. The "Class" tab selects a style (point descriptor) and size for the points. A palette of 256 colors is available to be assigned to the point elevation.

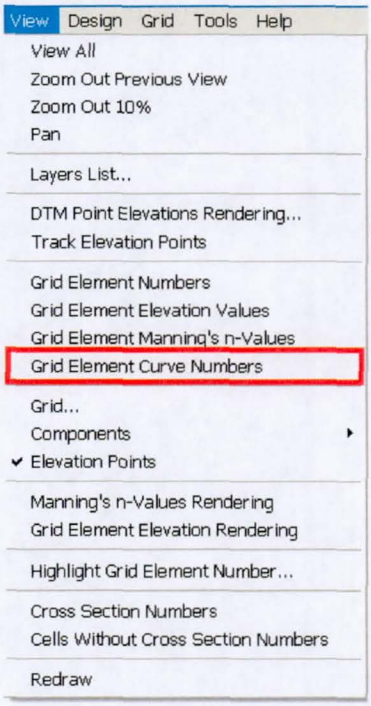


5.3.7 Track Elevation Points Command (View Menu)

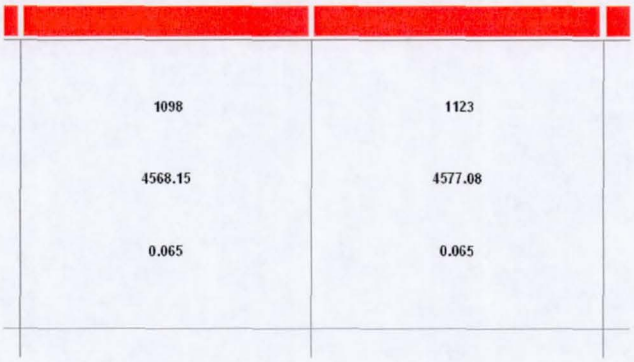


This command queries individual terrain elevation points and displays the data for the selected point in the toolbar point elevation box. The mouse cursor changes to the inquiry mode to show that this option is active.

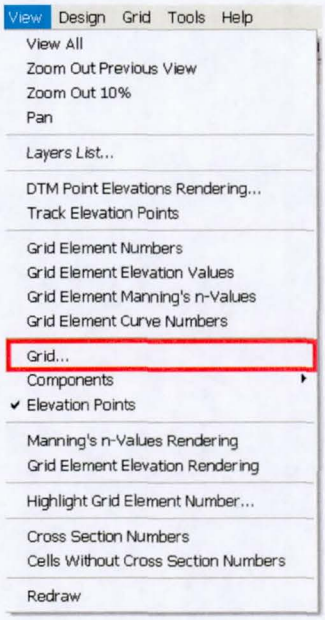
5.3.8 Grid Element Numbers, Elevations, and n-values and Curve Numbers Commands (View Menu)



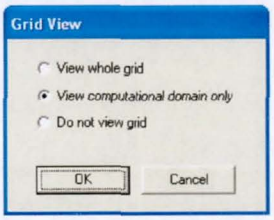
Use these commands to display grid element numbers, elevations, n-Manning's values or Curve Numbers inside the grid elements. Each command can be toggled and used jointly to display any of the three values. Note that if the grid system is large, the numbers may not fit into the elements. Zoom in to enlarge the view and to clearly see the element numbers. In each element, the first number is the grid element number, the second is the terrain elevation and the third the Manning's n value.



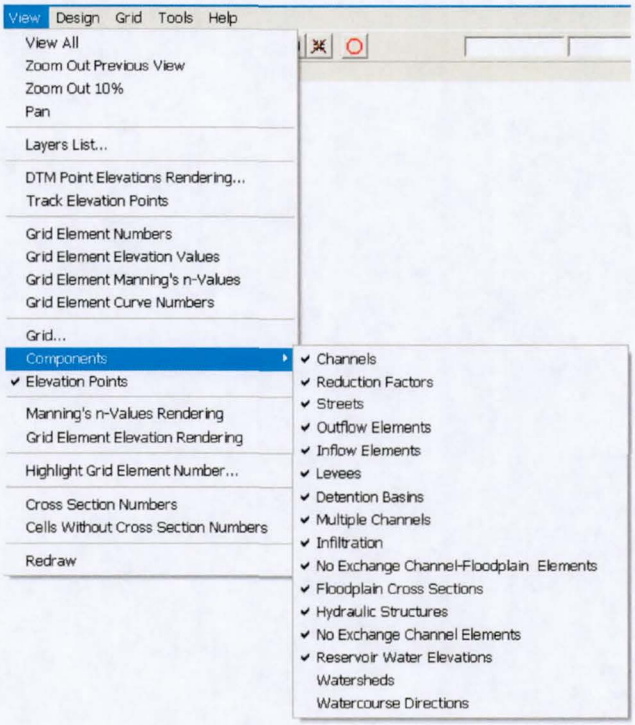
5.3.9 Grid...Command (View Menu)



This command customizes the grid display. You have the following options:

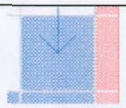
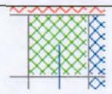
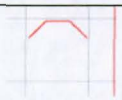
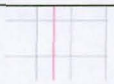


5.3.10 View Components Commands (View Menu)

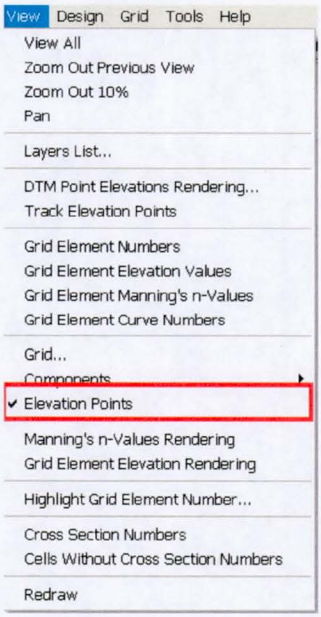


These commands enable you to view or hide *FLO-2D* components. You may mix and match the different components depending on the desired view.

These are some example views of the different *FLO-2D* components as represented in *GDS*:

Component	***GDS* Graphic Representation**
Channels	
Reduction Factors	
Streets	
Outflow Elements	
Inflow Elements	
Levees	
Detention Basins	
Multiple Channels	
Infiltration	
No Exchange Channel-Floodplain Elements	
Floodplain Cross Sections	
Hydraulic Structures	
No Exchange Channel Elements	
Watersheds	
Watercourse Directions	

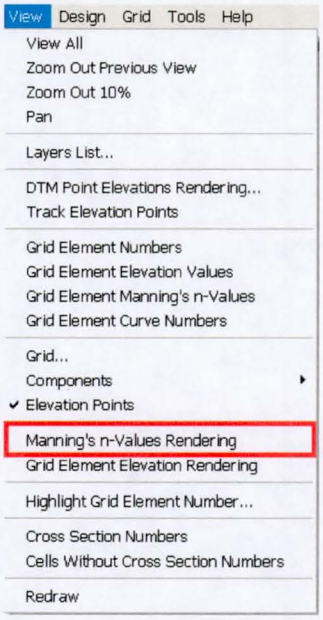
5.3.11 View Elevation Points Commands (View Menu)



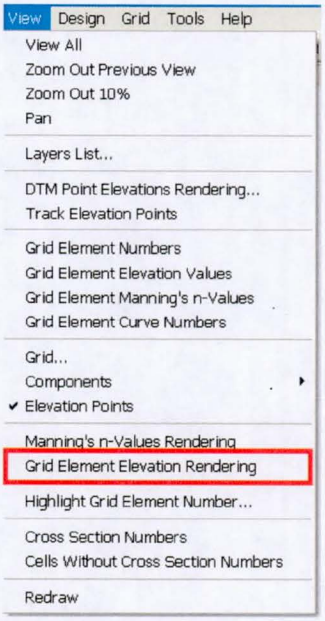
Displays elevation points assigning colors as a function of elevation.

5.3.12 Manning's n Values Rendering Command (View Menu)

Will display a colored hatched pattern on the elements according to the Manning's n value.

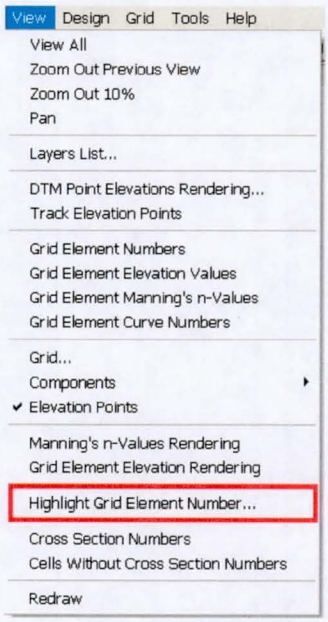


5.3.13 Grid Element Elevation Rendering Command (View Menu)

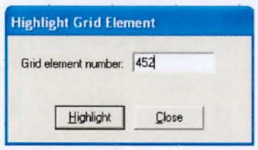


Plots colored grid elements according to the elevations.

5.3.14 Highlight Grid Element Number… (View Menu)



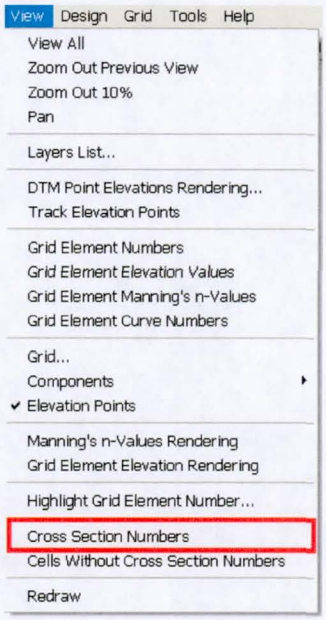
Enter a grid element number in the following dialog box to locate it in the *FLO-2D* grid.



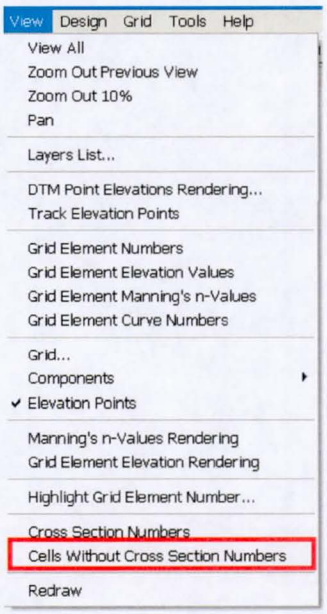
When you click the *Highlight* button, the grid element will blink to identify it. Note that if the selected element number is not in the current view, you may have to zoom out to see it.

5.3.15 Cross Section Numbers… (View Menu)

This command displays cross section numbers for Natural Channels.

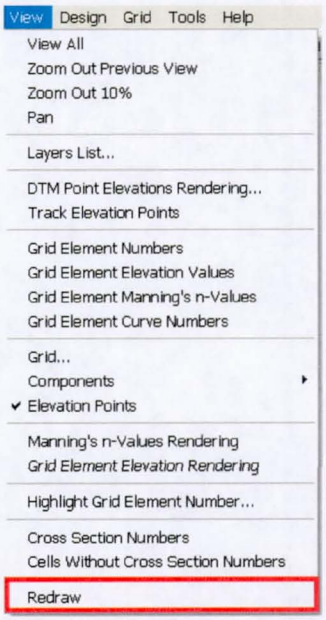


5.3.16 Cells Without Cross Section Numbers… (View Menu)



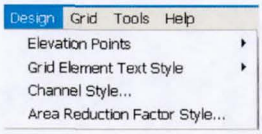
This command displays natural channel grid elements that do not have any assigned cross section.

5.3.17 Redraw Command (View Menu)

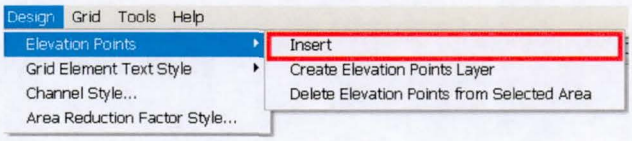


Use this command to redraw the visible objects in the working region.

5.4 Design Menu Commands:

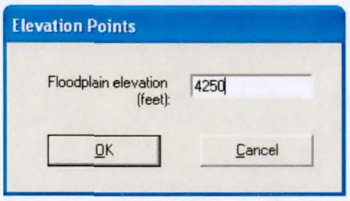


5.4.1 Elevation Points/Insert Command (Design Menu)



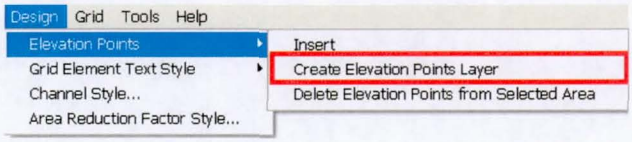
Use this command (or the icon) to insert elevation data at selected points within the working region. To use this tool:

1. Select the *Elevation Points/Insert* command (Design menu) or the toolbar icon and click the left mouse button.
2. A dialog box appears for the elevation entry in feet or meters. Click *'OK'* to accept the value.



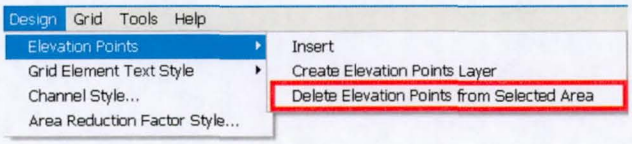
3. Click a point on the working region where you want to assign this elevation data.
4. Repeat step 3 as many times as needed.

5.4.2 Elevation Points/Create Elevation Points Layer Command (Design Menu)



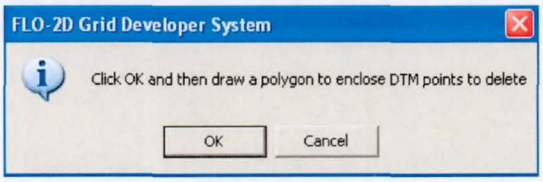
To optimize display times, *GDS* does not automatically create the Elevation layer. Use this command to create the elevation point layer.

5.4.3 Elevation Points/Delete Elevation Points from Selected Area Command (Design Menu)



This command deletes elevation data points within the working region. To use this tool:

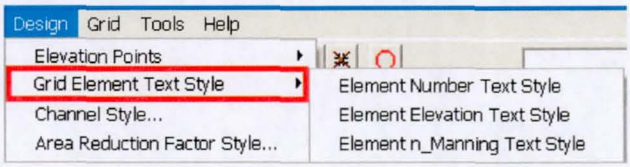
1. Select the *Elevation Points/ Delete Elevation Points from Selected Area* command (Design menu) and click the left mouse button.
2. The following dialog box appears:



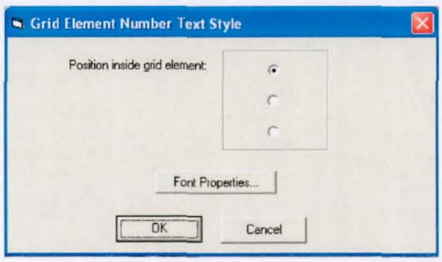
3. Draw a polygon ending with a double click to close the polygon.

4. Click *'OK'* to delete the points inside the polygon.
5. To save the edited DTM points file, click on the Save Elevation Points command:

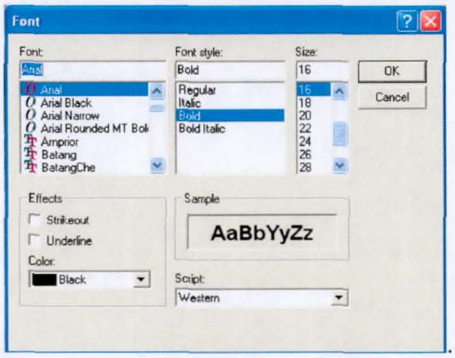
5.4.4 Grid Element Text Style Commands (Design Menu)



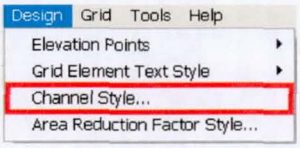
Use these commands to edit the text styles for the grid element number, elevation, or Manning's n-value.



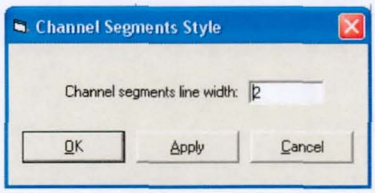
You can set the relative position of the number in the square grid element (upper, middle or down positions). The Font Properties… button allows you to change, font type, style, size, etc.



5.4.5 Channel Style (Design Menu)

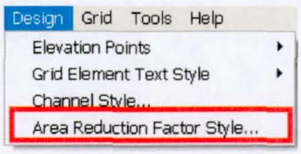


This command will change the line width used to represent channels. The *GDS* displays this dialog box to set the line width:

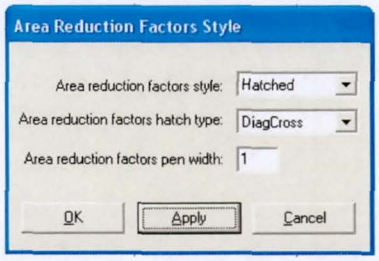


Click on *Apply* and then '*OK*' to change to the selected channel line width display

5.4.6 Area Reduction Factor Style (Design Menu)



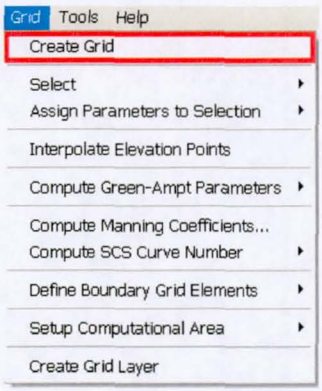
To change the display of area reduction factor in a grid element, chose from Solid, Hollow (with boundary display of different pen widths) or Hatched rendering with various hatching options.



Click on *Apply* and then *'OK'* to change to the selected Style.

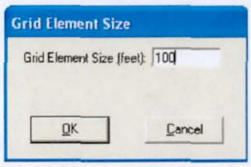
5.5 Grid Menu Commands:

5.5.1 Create Grid Command (Grid Menu)



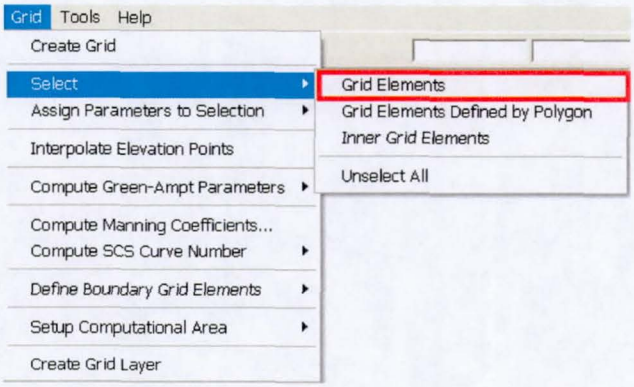
Use this command to create the grid system template of square elements for the *FLO-2D* model. To use the *Create Grid* command:

1. Select the *Create Grid* command (Grid menu).
2. A dialog box appears requesting the square grid element size or side length (ft or m):



3. Select *'OK'* to accept the value.
4. The *GDS* system will automatically overlay a grid template that is centered in the working region. The grid elements will be centered on each node.

5.5.2 Select/Grid Element Command (Grid Menu)

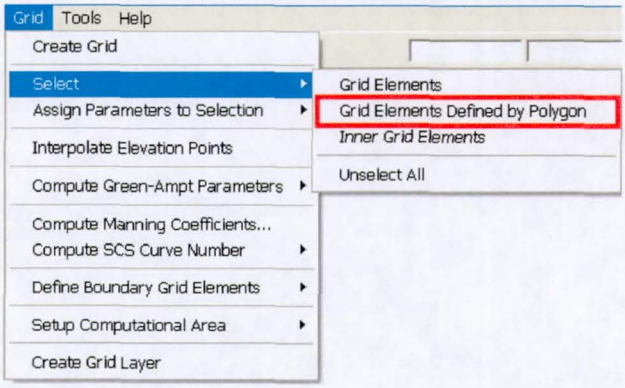


Use this command (or the # icon in the tool bar) to select one or more grid elements. The selected elements are shown as cross-hatched with diagonal lines. With the *Assign Parameters to Selection* command, you can assign attribute values to the selected grid elements. To use the *Select / Grid element* command:

1. Choose the *Select/Grid element* command (Grid menu).
2. The cursor changes to a cross.
3. Mouse click on the grid element you want to select.
4. Repeat step 3 for each selected element.

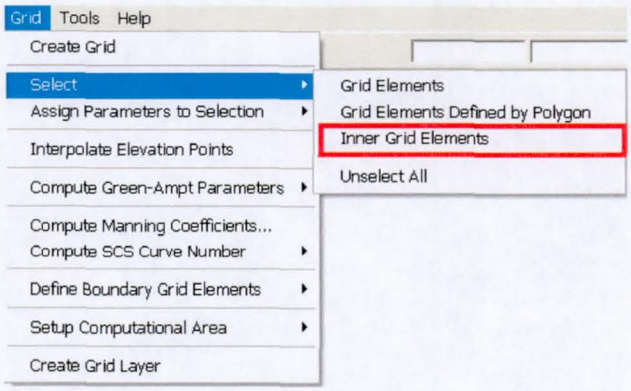
To unselect previously selected elements, simply click on them again. To select a group of elements, press the *Shift* key simultaneously with the left mouse button and drag the mouse pointer over the desired elements. To unselect a group of elements press the *Control* key and the left mouse button, then drag the mouse pointer over the elements you want to unselect. When dragging the mouse over the grid elements, they are painted to indicate your selection. After a grid element or group of elements is selected, you may use the *Assign Parameters to Selection* command to assign various attributes to the selected elements.

5.5.3 Select/Grid Elements Defined by Polygon Command (Grid Menu)



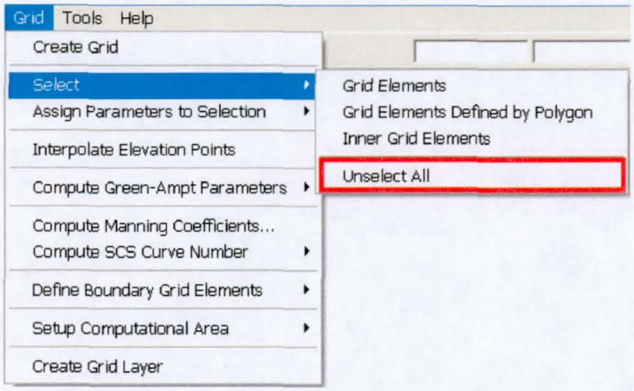
This command (or the icon in the tool bar) will select all the grid elements within a user defined polygon. Attributes can then be assigned to the selected elements using the *Assign Parameters to Selection* command.

5.5.4 Select/Inner Grid Elements Command (Grid Menu)



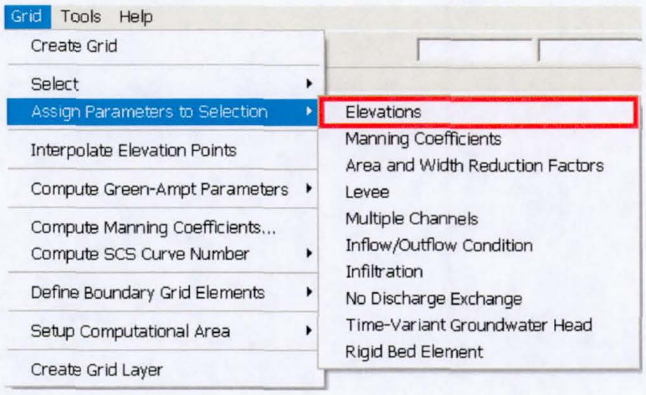
This command will select all the grid elements within the computational domain and hatch them with diagonal lines. Use the *Assign Parameters to Selection* command to assign various attributes to the selected elements.

5.5.5 Select/Unselect All Command (Grid Menu)

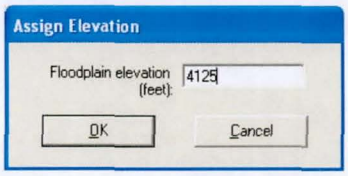


This command will unselect all the elements previously selected with the *Select* command. The toolbar icon will also perform this function.

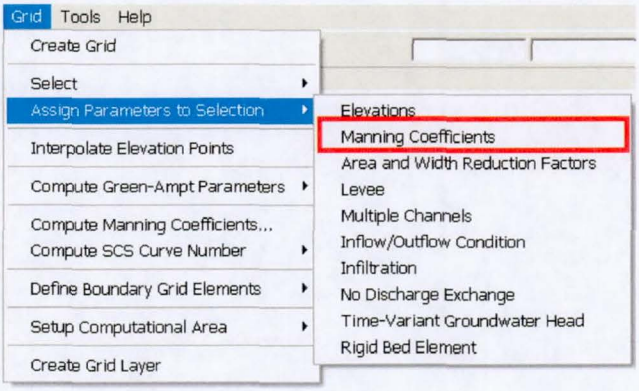
5.5.6 Assign Parameters to Selection/Elevations Command (Grid Menu)



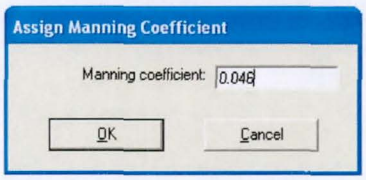
With this command you can assign an elevation to grid elements that have been selected using the *Select* command. Upon selection, a dialog box appears prompting you to enter an e elevation. Click *'OK'* to assign the elevation to all the selected elements:



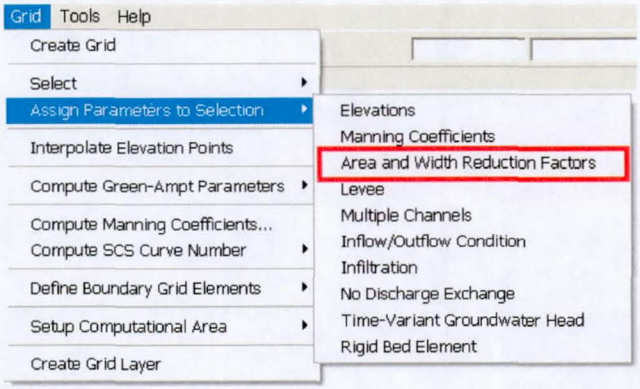
5.5.7 Assign Parameters to Selection/Manning's Coefficient Command (Grid Menu)



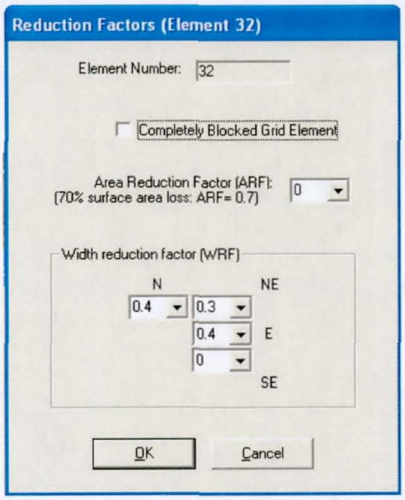
A Manning's roughness coefficient (n-value) is assigned to the grid elements previously selected with the *Select* command. A dialog box appears prompting you to enter the n-value:



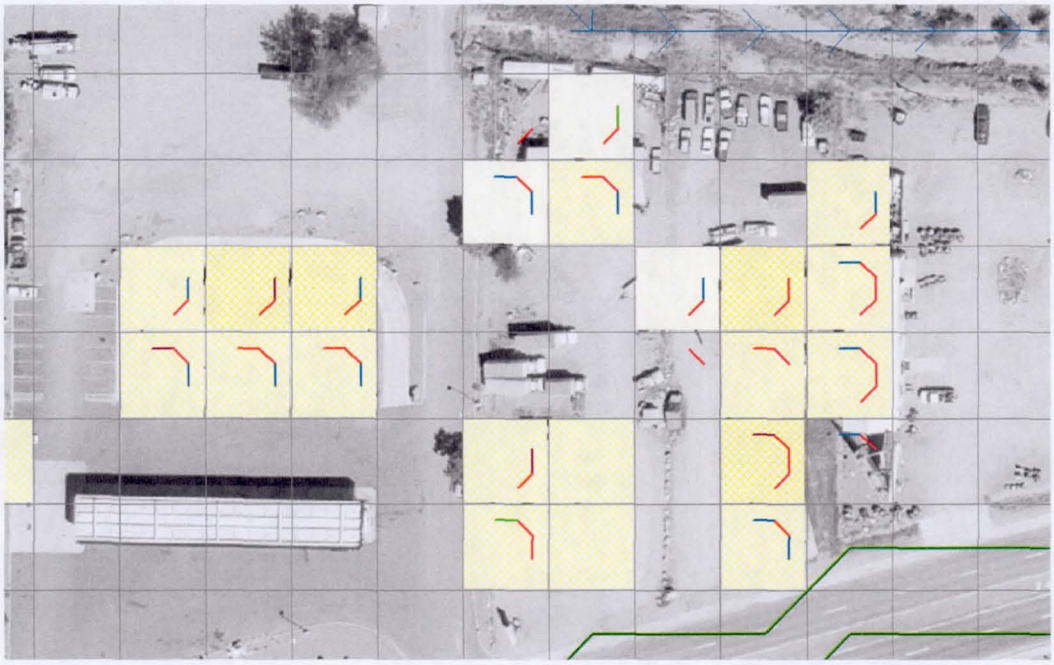
5.5.8 Assign Parameters to Selection/Area and With Reduction Factors Command (Grid Menu)



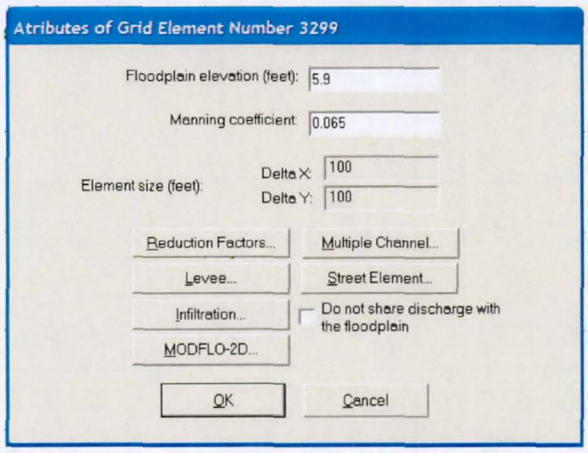
This command is used to assign area and width reduction attributes to the selected grid elements. A dialog box appears prompting you to enter the Area Reduction Factor (ARF) and the Width Reduction (WRF) factors:



After clicking *'OK'*, the assigned ARF and WRF values will appear graphically as follows:

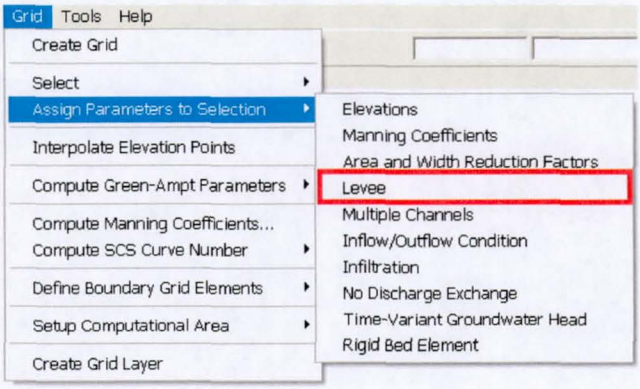


The colored cells indicate varying levels of storage loss (ARF values). The colored lines reflect the varying levels of WRF flow direction blockage. Only four of the eight potential directions (N, E, NE, SE) for each grid element need to be assigned to identify complete blockage. The other 4 directions are assigned to contiguous grid elements. It is not necessary to identify the blockage in both directions (e.g. NE and SW). The user may double click on any cell to change the area or width reduction factors. The following dialog box appears:

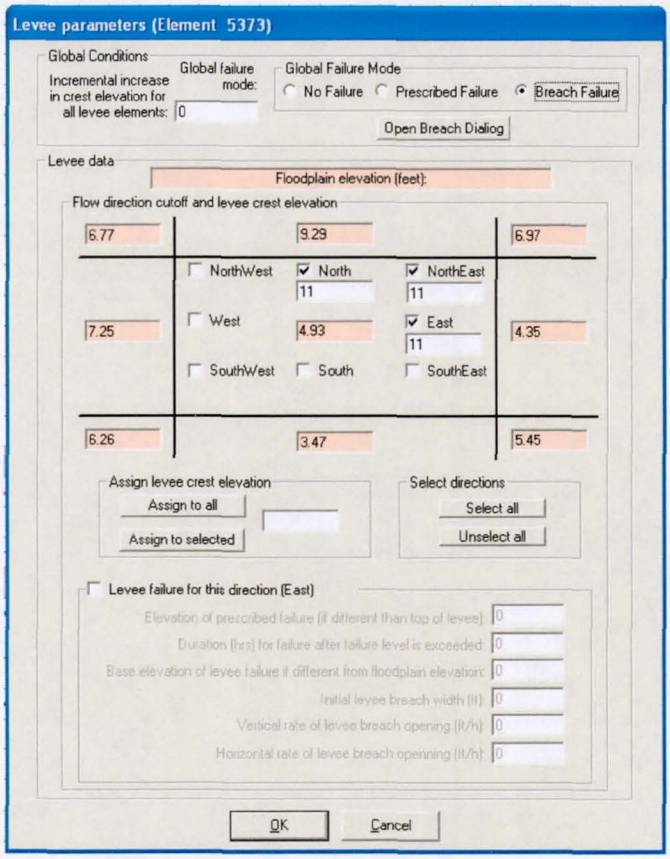


Click on *Reduction Factors* button to change the factors using this dialog box. It should be noted that each grid element can share discharge in eight directions. The grid element can be considered to be an octagon and each WRF factor refers to the percent blockage of one of the eight sides. The maximum ARF value is 0.95 unless the grid element is totally blocked (Check box *Completely Blocked Cell*). This insures that at least 5% of the grid element is left for flow storage. For ARF values greater than 0.95, the model will reset the ARF to 0.95. To totally block out a grid element with zero storage, check the box *Completely Blocked Cell* in the dialog box.

5.5.9 Assign Parameters to Selection/Levee Command (Grid Menu)



Use this command to assign levee and levee failure parameters to the selected grid elements. A dialog box will appear with the grid element floodplain elevation across the levee in that flow direction for reference. When you click on a check button to set up a levee for each selected direction (NW, N and NE in the below dialog box), a text box will appear just below each direction to input the levee crest elevation. Note that the levee crest elevation must be higher than that of all surrounding elements floodplain elevation. Obviously, it isn't necessary to specify a levee in a given flow direction is the levee crest elevation is lower than contiguous grid element floodplain elevation.



Each data entry is described below:

Incremental increase in crest elevation for all the levee nodes (RAISLEV)

Global incremental increase in levee crest elevation (ft or m) for all the levee grid elements.

Global Failure Mode provides two modes of failure:

Prescribed failure where you may provide predetermined breach data in the bottom part of the dialog, or

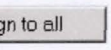
Breach failure where the model simulates the breach erosion from overtopping or piping. In this case, breach parameters should be entered clicking the Open Breach Dialog button.

Flow direction cutoff

Check buttons to define the flow direction (of the 8 possible overland flow directions) that will be cutoff by a levee.

Levee Crest Elevation (LEVELEV)

Elevation (ft or m) of the top of the levee.

Assign to all button 

Assigns the cutoff flow direction and levee crest elevation to all selected grid elements.

Levee failure for this direction box ☑ Levee failure for this direction

Enables input parameters to be assigned for levee prescribed failure modeling for the selected direction.

Elevation of prescribed failure

Assign the maximum water surface elevation at failure (FAILEVEL) if is different than the levee crest (LEVELEV). This enables the levee to fail prior to being overtopped. Set the FAILEVEL variable to zero to simulate levee failure when it is overtopped.

Duration (hrs) for failure after failure level is exceeded (FAILTIME)

Duration in hours until levee failure after the FAILEVEL elevation is exceeded by the flow depth. Set this variable to zero if the level fails immediately when overtopped or when FAILEVEL is exceeded.

Base elevation of levee failure if different from floodplain elevation (LEVBASE)

The final elevation of the levee after failure is completed. This enables the levee to fail to an elevation that is different from the floodplain elevation. Set this variable to zero if levee failure results in the complete levee failure to the floodplain elevation.

Initial levee breach width (FAILWIDTH)

The initial flow width (ft or m) of levee failure. This flow width relates to one of the eight flow directions and should be less than the length of an octagon side (length of the side of a grid element x 0.4142).

Vertical rate of levee breach opening (FAILRATE)

The rate of vertical levee failure (ft/hr or m/hr).

Horizontal rate of levee breach opening (FAILWIDRATE)

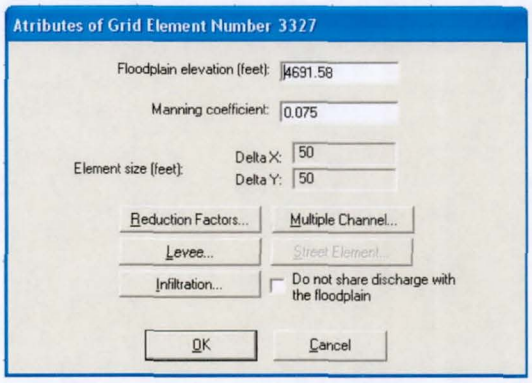
The rate at which the levee breach widens (ft/hr or m/hr). The breach stops increasing if the breach exceeds the grid element width for that direction.

The following rules govern the levee data base:

1. The rules for the levee failure are:
   a. If the levee fails when overtopped by the flow, set FAILEVEL and FAILTIME = 0.
   b. If the levee is to fail at specified level below the top of the levee, set FAILEVEL equal to the failure elevation.
   c. If the levee is to fail at a specific level below the top of the levee after being inundation for a specific duration, assign appropriate values to both the FAILEVEL and FAILTIME variables.
   d. If the levee fails to a new base levee that is different than the floodplain elevation, assign an appropriate elevation to LEVBASE.
   e. After levee failure, flow will discharge through a breach width of FAILWIDTH for that flow direction.
   f. Progressive levee failure is simulated by assigning a value to either FAILRATE (ft/hr or m/hr) FAILWIDRATE (ft/hr or m/hr) or both. A new levee crest elevation or failure width is computed as failure proceeds. FAILRATE is a vertical rate of decrease in the levee breach elevation.

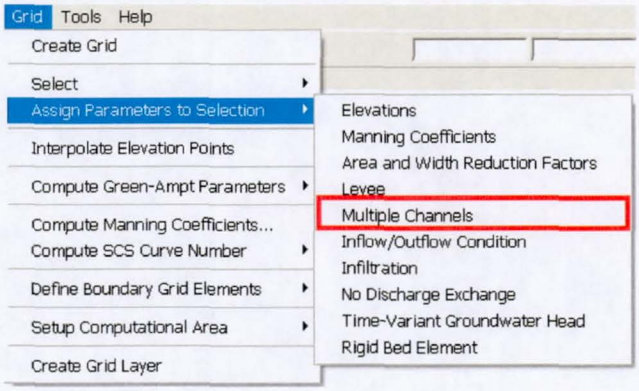
2. No multiple channels will be assigned to grid elements with levees. Multiple channels in a levee grid element are eliminated automatically by the model.
3. Each levee grid element can have up to eight failure directions. Note that the failure with will be based on an octagon with is 0.4142 times the side length of the grid.
4. Flow over a levee is computed as broad crested weir flow using a coefficient of 2.85 until the tail-water depth is 85% of the headwater depth at which point the flow reverts to overland flow based on the water surface elevations on each side of the levee and the flow depth over the levee.

The user may also double click on any cell to edit the levee data and the following dialog box appears:

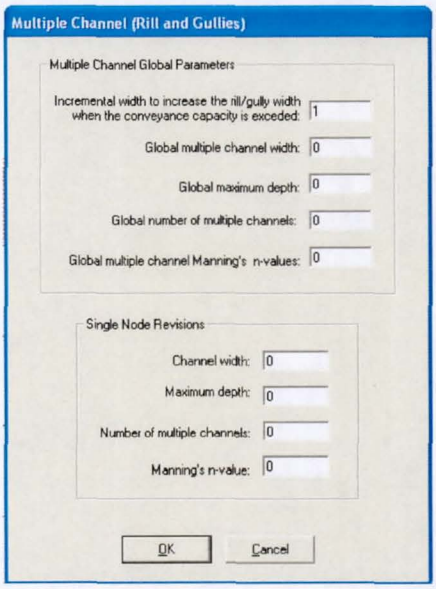


Click on the *Levee* button to access the levee dialog box.

5.5.10 Assign Parameters to Selection/Multiple Channels Command (Grid Menu)



This command is used to define multiple channels that simulate rill and gully flow on the floodplain. For this component, concentrated rill and gully flow (flow in rectangular channel) rather than overland sheet flow will be simulated to route the flow between designated floodplain grid elements. The following dialog box is used to input the multiple channel data:



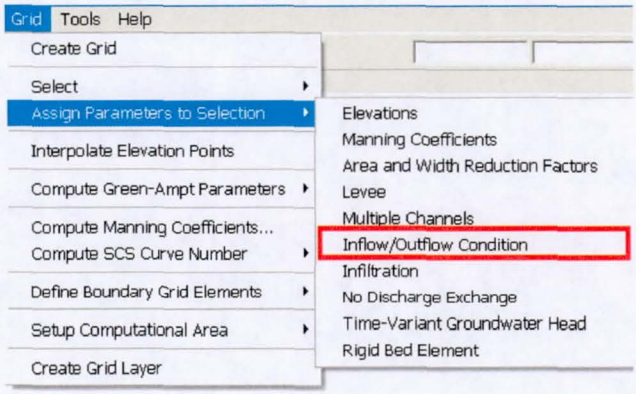
The upper group (*Multiple Channel Global Parameters)* consists of global parameters for the assignment of multiple channel attributes including the incremental width increase, multiple channel width and depth, number of channels per grid element, and n-values. The second group (*Single Node Revisions)* of parameters is for the assignment of multiple channels to individual grid elements.

If a gully fills with water and is about to overflow, it is assumed that the gully is an alluvial channel and will widen to accept more flow. Thus when the flow depth exceeds the assigned maximum gully channel depth, the model increases the gully width by the incremental width. Once the flow is in a gully, the gully will not overflow back onto the floodplain, but rather will continue to widen until the gully is wider than the grid element or until there is no more overland storage area left in the grid element. The flow routing will then revert to overland sheet flow in that element. The following rules governing the assignment of the multiple channel data:

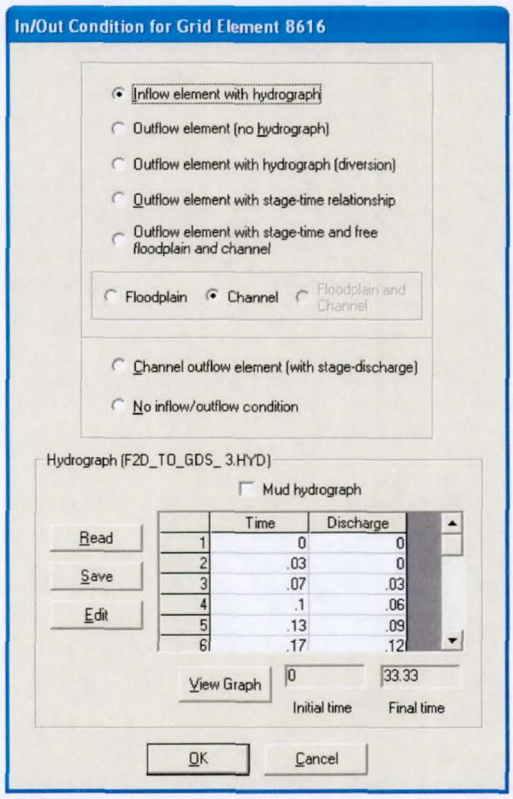
1. When the flow depth exceeds the multiple channel (gully) depth, the flow width of the gully is increased by the incremental width.
2. If it is desired to force the flow to stay in a channel of fixed width, set the incremental width equal to zero.
3. The total flow width is determined by multiplying the number of channels in each grid element by the appropriate width.
4. If the number of multiple channels assigned in a grid element is set equal to zero, then the overland sheet flow without multiple channels is assumed.
5. The values given in the single grid element data group will supersede those in the global parameter group.
6. If a grid element is assigned multiple channels and, in addition, contains a main channel or building (ARF value), such that the available floodplain surface storage area is less than 50% of the original grid element surface area, then the model will reset that multiple grid element to overland sheet flow (i.e. no multiple channels). Multiple channel flow will not be simulated in the grid elements with streets and the model will automatically eliminate any multiple channels in grid elements with streets. The available surface area

in a grid element can be reviewed in the SURFAREA.OUT output file.

5.5.11 Assign Parameters to Selection/Inflow/Outflow Condition Command (Grid Menu)

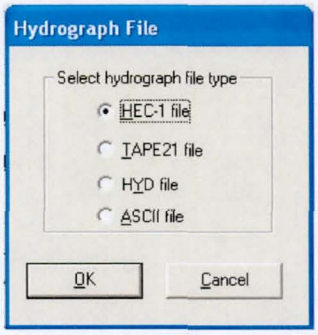


Use this command to define inflow and outflow elements in selected grid elements. The flowing dialog box allows editing these boundary conditions:

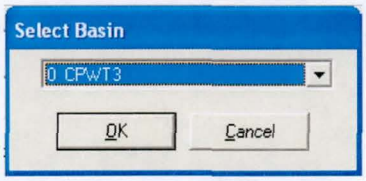


5.5.11.1 Setting inflow nodes

Clicking the first radio button will assign an inflow hydrograph to a grid element. If there is a channel in the selected element, you can assign the hydrograph to either the channel or the floodplain. When the user selects the radio button *'Inflow element with hydrograph'*, the *'Hydrograph'* data group is activated. The "*Read*" button displays a dialog box to import a hydrograph with HEC-1, Tape21, HYD or ASCII files formats:



The HEC-1 file option will display all the hydrographs at basin concentration points in a Corps of Engineers HEC-1 hydrologic model output file. The user can select an inflow hydrograph from the HEC-1 file:

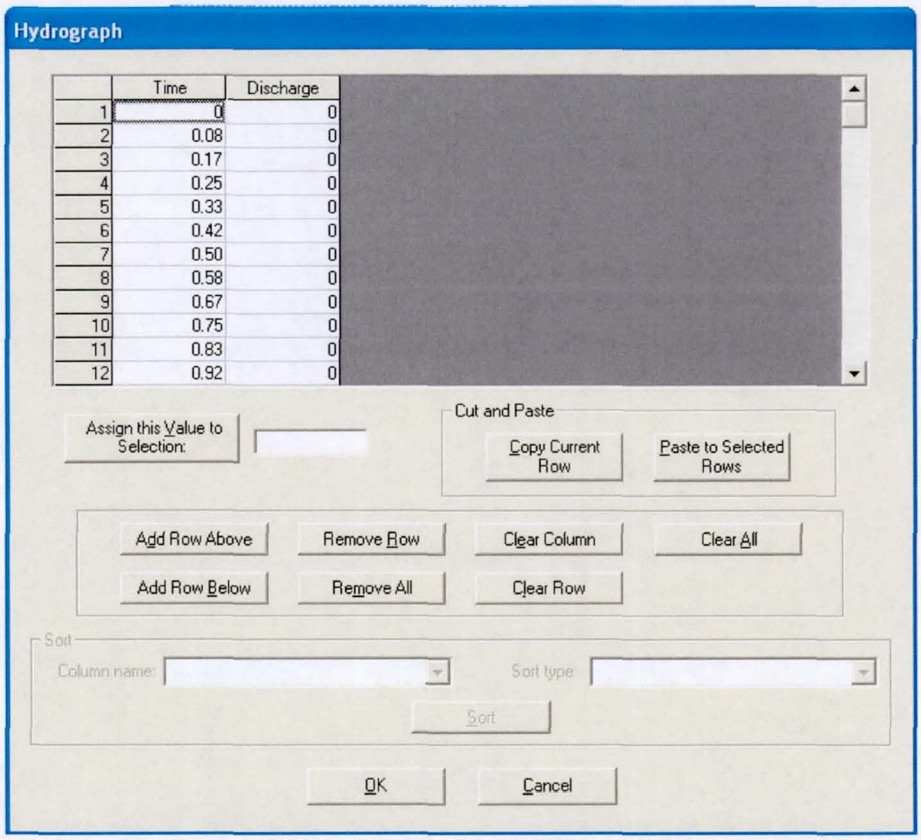


After selecting the HEC-1 hydrograph and clicking *'OK'*, the hydrograph data are loaded into the *Hydrograph* group table in the above dialog box.

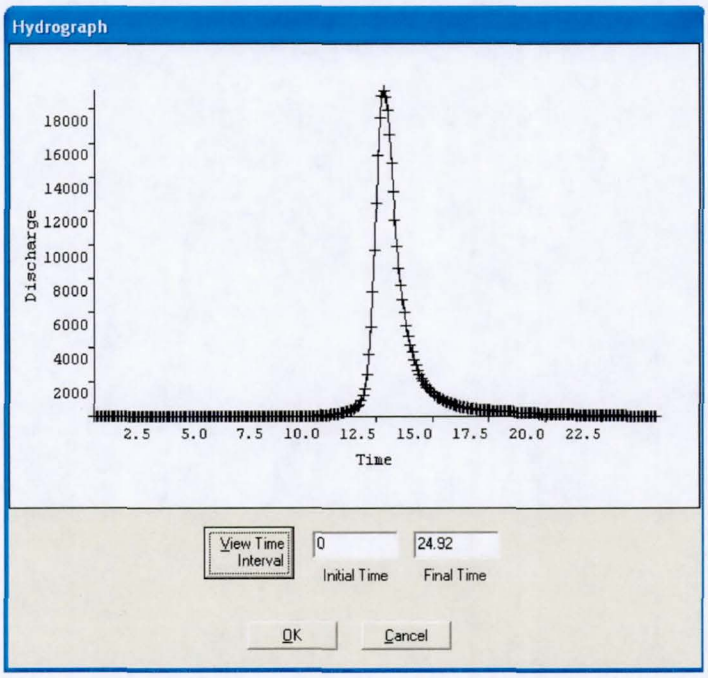
*.HYD files are ASCII files generated by the *GDS* to save the hydrograph data. They are created with the "*Save Table*" button. The first line contains the initial and final time of the hydrograph selected with the "Select Time Interval" button. The time and discharge discretized hydrograph pairs follow in two columns of data. These files can be used to:

- Recover the hydrographs when a project is read from a TOP file;
- Redefine the project area when the INFLOW.DAT becomes obsolete because the grid element numbers change. The *.HYD files enable the hydrographs to be recovered for assignment to new grid elements.
- *.HYD files permanently store hydrographs that could be imported to other projects or grid elements in the same project.

The "*Edit Table*" button can be used to edit hydrograph values, insert rows, delete rows, or sort rows in ascending order (time column). This button a displays an editor dialog box:



The "*View Hydrograph*" button of the above *In/Out Condition* dialog box plots the hydrograph in the following window:



The user may select the initial and final time for the inflow hydrograph. For example, selecting *Initial Time* = 10.0 hours and *Final Time* = 16.0 will redefine the hydrograph limits and in this eliminate a number of unnecessary zero values on the rising and falling limbs of the hydrograph.

Inflow grid elements with assigned hydrographs are displayed in *GDS* with a distinct color for identification. When the user selects "*No Inflow/Outflow condition*" and clicks *'OK'*, the grid element recovers its original color to indicate the absence of an inflow or outflow node. Inflow nodes and the linked hydrographs will be written to the INFLOW.DAT file. Outflow nodes are assigned though the option "*Outflow element (no hydrograph)*". The list of outflow nodes will be written to the OUTFLOW.DAT file.

Initial Assignment of Hydrographs

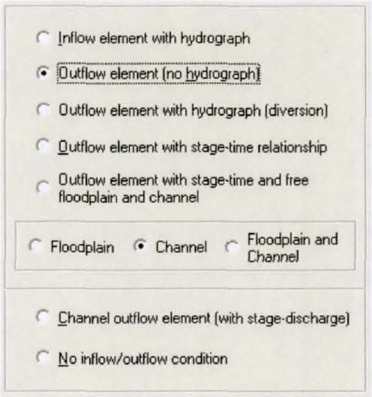
The initial assignment of hydrographs to inflow nodes has three options:

1. ***Project is created from FLO-2D project*** (*GDS* command "*File. New Project from FLO-2D Project*"). In this case INFLOW.DAT is used by the *GDS* system to assign hydrographs to pre-selected grid elements.
2. ***Project is read from TOP*** (*GDS* command "*File. Open Project*"). The new TOP will include references to *.HYD files that are related to grid elements defined by the user.
3. All other methods of creating a project. The user will be required to create the grid system and then later right-click on the selected inflow nodes to assign hydrographs as described above.

5.5.11.2 Setting outflow nodes

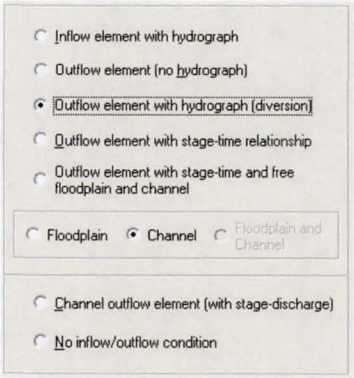
These are the options for outflow nodes:

Outflow element (no hydrograph): This option can be assigned to the floodplain, to the channel (if there is a channel in the selected element), or to the combined floodplain and channel.

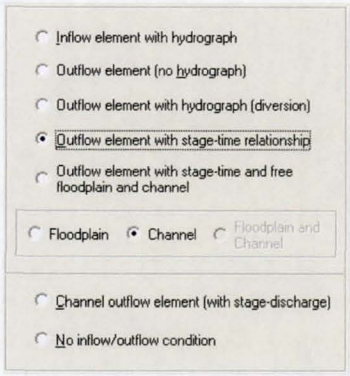


For this outflow assignment, the outflow nodes discharge all the inflow to them off the grid system using an approximate normal depth flow condition. The outflow node is essentially a sink.

Outflow element with hydrograph (diversion): Use this option to model an outflow hydrograph from the grid element.

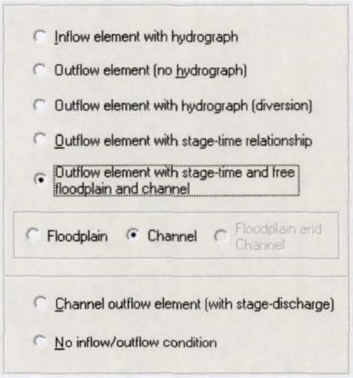


Outflow element with stage-time relationship: This grid element condition applies only to floodplain or channel but not both.



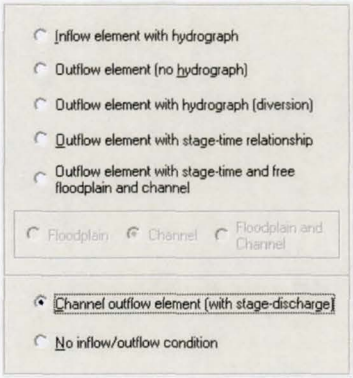
Use this water surface control to simulate flooding from storm surges/tsunamis or any type of water surface elevation control such as tidal effects or temporally variable backwater conditions. For example, you can model the flooding in a coastal area produced by storms surges generated by tropical storms, hurricanes or tsunamis where the ocean stage rise and fall has a limited duration. This condition can be assigned to nodes along the coastal boundary to simulate ocean flooding in an urban area. The grid element with a stage-time relationship is not assigned as an outflow node and does not have to be at a boundary. This condition can also be used to assign a downstream water surface control that might be required for a FEMA Flood Insurance Study.

Outflow element with stage-time and free floodplain and channel:



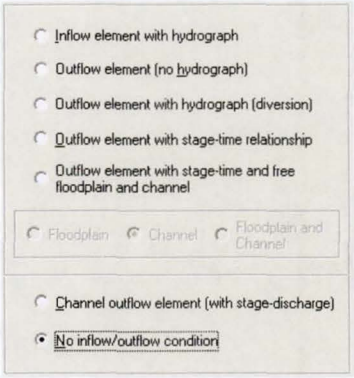
This outflow condition is a combination of the stage-time relationship and outflow node for the either the floodplain or channel. In this case, the water surface elevation is temporally controlled by the stage-time relationship, but any inflow to the floodplain or channel is discharged as an outflow node. This enables an overland storm surge or tsunami wave to drain back to the ocean in the same grid element when the surge wave is retreating.

Channel outflow element (with stage-discharge):



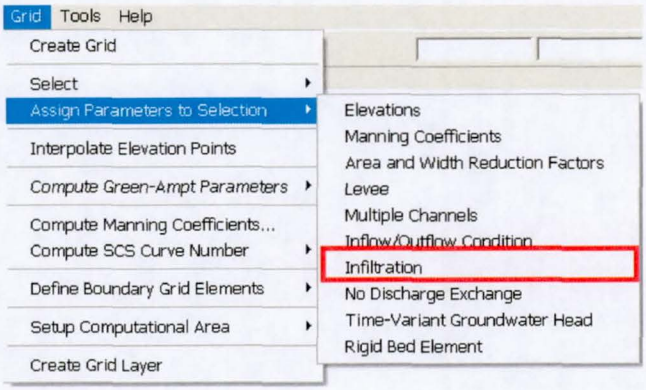
Use this option to assign a stage-discharge relationship to the channel. In this case, the channel outflow discharge will be controlled by assigned stage discharge pairs to simulate a weir, culvert or other water surface hydraulic control.

5.5.11.3 No inflow/outflow condition

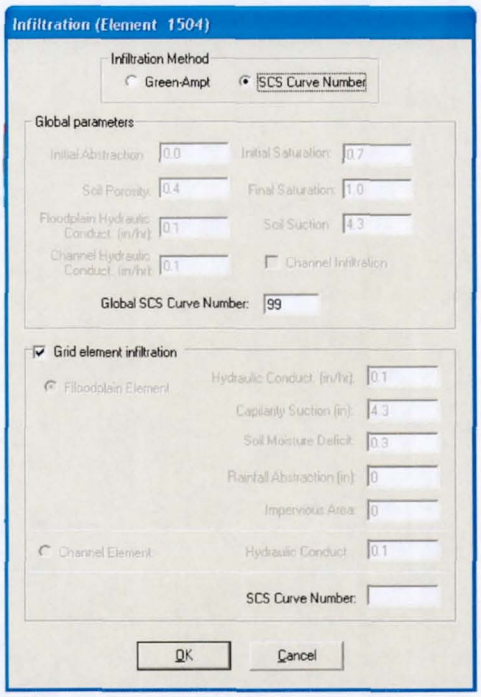


Use this option to clear all inflow or outflow boundary conditions in the selected grid element. In this case, there is no assignment of inflow or outflow for the grid element.

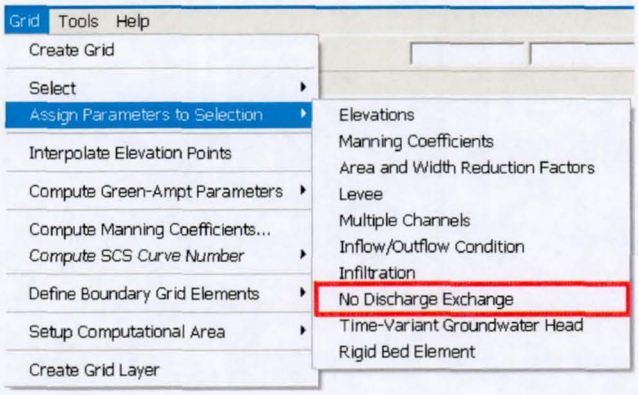
5.5.12 Assign Parameters to Selection/Infiltration Command (Grid Menu)



Use this option to assign infiltration data to the selected grid elements in the following dialog box:

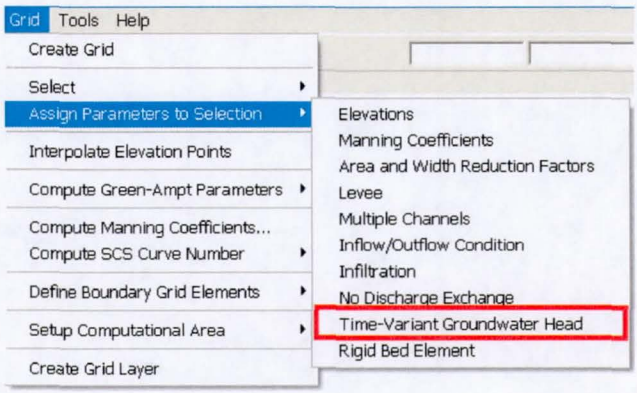


5.5.13 Assign Parameters to Selection/No Discharge Exchange Command (Grid Menu)



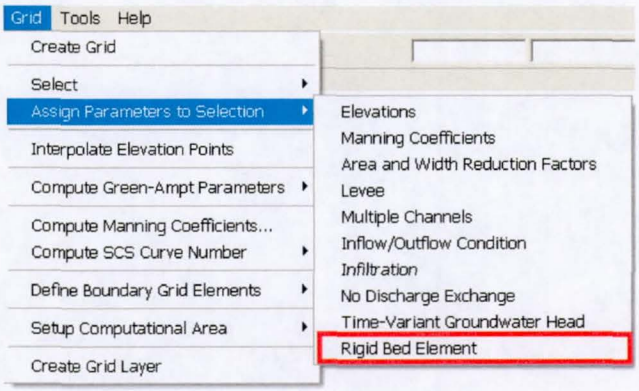
Use this command to identify channel elements that will not exchange discharge with the floodplain (e.g. channel culvert or enclosed channel where no floodplain inflow can occur).

5.5.14 Assign Parameters to Selection/Time-Variant Groundwater Head Command (Grid Menu)



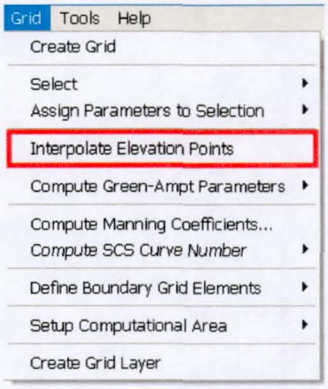
Use this command when using the *MODFLOW* groundwater flow model link to impose a time series of groundwater head at the selected nodes.

5.5.15 Assign Parameters to Selection/Rigid Bed Element Command (Grid Menu)



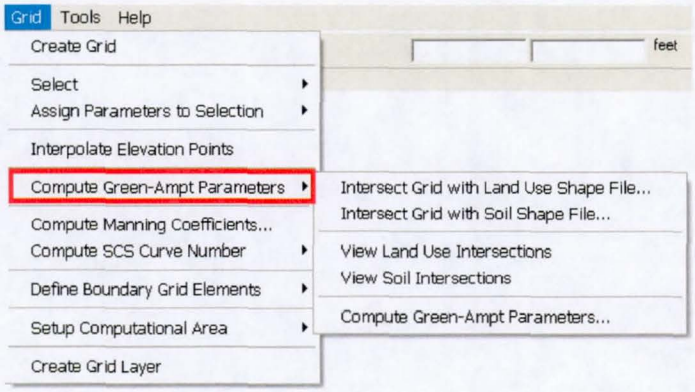
Use this command to setup elements that do not scour during the sediment transport simulation.

5.5.16 Interpolate Elevation Points Command (Grid Menu)



Use this command to interpolate and assign all the grid element elevations in the computational domain. The interpolated elevations are based on the imported DTM points or the elevation points that are added to the working region. The interpolation options are selected in the *Options/Interpolation* command of the *Tools* menu.

5.5.17 Compute Green-Ampt Parameters Command (Grid Menu)



This command is the starting point for the Green-Ampt parameter calculations. There are several steps that have to be completed in a prescribed sequence. The final result is the creation of the INFIL.DAT file containing the Green-Ampt infiltration parameters. To initiate this procedure, you have to load a landuse shape file and a soil shape file and the corresponding soil and landuse tables must be available. In addition, the *FLO-2D* grid system data files must exist. The general procedure is as follows:

1. Read soil shape file.
2. Read landuse shape file.
3. Determine soil type polygon intersections with the *FLO-2D* grid elements.
4. Determine landuse polygon intersections with the *FLO-2D* grid elements.
5. For each grid element compute hydraulic conductivity (*XKSAT)* average.

$$\overline{XKSAT} = ALOG\left(\frac{\sum A_i \log(XKSAT_i)}{A_{GE}}\right)$$

Where:

$XKSAT_i$ is obtained from the soil table and
A_i is subarea intercepted by the grid element from the 3rd column of the landuse table and A_{GE} the grid element area.

6. For each grid element, compute wetting front capillary suction PSIF according to the following regressions as a function of *XKSAT* (Generated from Figure 4.3 of the Maricopa County Drainage Design Manual, Volume I).

XKSAT (in/hr)	PSIF (in)
$0.01 \leq XKSAT \leq 1.2$	$PSIF=EXP(0.9813-0.439*Ln(XKSAT)+0.0051(Ln(xksat))^2+0.0060(Ln(XKSAT))^3)$

7. For each grid element, compute volumetric soil moisture deficiency *(DTHETA)* according to the following table. The specific table used for DTHETA depends on the *saturation* field of the soil table (6th column).

Saturation = DRY

XKSAT (in/hr)	DTHETA DRY
$0.01 \leq XKSAT \leq 0.15$	DTHETA =EXP(-0.2394 + 0.3616 Ln(XKSAT))
$0.15 < XKSAT \leq 0.25$	DTHETA =EXP(-1.4122 - 0.2614 Ln(XKSAT))
$0.25 < XKSAT \leq 1.2$	DTHETA = 0.35

Saturation = NORMAL

XKSAT (in/hr)	DTHETA NORMAL
$0.01 \leq XKSAT \leq 0.02$	DTHETA = EXP(1.6094 + Ln(XKSAT))
$0.02 < XKSAT \leq 0.04$	DTHETA = EXP(-0.0142 + 0.5850 Ln(XKSAT))
$0.04 < XKSAT \leq 0.1$	DTHETA = 0.15
$0.1 < XKSAT \leq 0.15$	DTHETA = EXP(1.0038 + 1.2599 Ln(XKSAT))
$0.15 < XKSAT \leq 0.4$	DTHETA = 0.25
$0.4 < XKSAT \leq 1.2$	DTHETA = EXP(-1.2342 + 0.1660 Ln(XKSAT))

Saturation = WET or SATURATED

DTHETA = 0 for all XKSAT

8. Adjust *XKSAT* (computed in step No. 1) as a function of the vegetation cover VC from the 5th field of the landuse table when XSAT < 0.4 in/hr. This requires a computation of the ratio the hydraulic conductivity for the vegetative cover to the bare ground hydraulic conductivity (C_K):

$$C_K = \frac{VC_K - 10}{90} + 1$$

$$\text{XKSATC} = \text{XKSAT} \sum_k \text{P}_\text{k} C_k$$

Where:

P_k is the percentage of the area within the grid element corresponding to C_k and XKSATC for each grid element is written to the INFIL.DAT file.

9. For each grid element compute the initial abstraction *IABSTR*:

$$IABSTR = \left(\frac{\sum A_i (IA_i)}{A_{GE}} \right)$$

Where:

IA_i is the initial abstraction in the subarea A_i intercepted by the element and is based

on the 3rd column of the landuse table;
The intercepted subareas are computed using the land use shape file and *IABSTR* is added to the INFIL.DAT file for each element.

Compute effective impervious area (%) for each grid element *(RTIMP)*.

$$RTIMP_1 = \left(\frac{\sum A_i (RTIMPS * EFF)_i}{A_{GE}} \right)$$

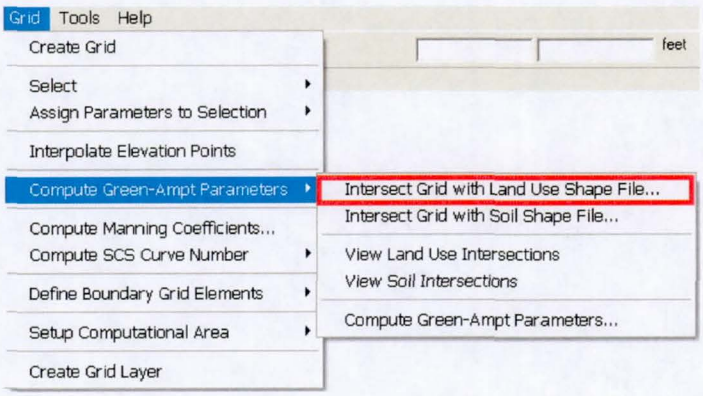
Where:

Ai is determined from the soil shape file;
A_{GE} is the grid element area, effective impervious area *EFF* is obtained from the 5th column of the soil table and
RIMPS is the percent rock outcrop obtained from the 4th column of the soil table.

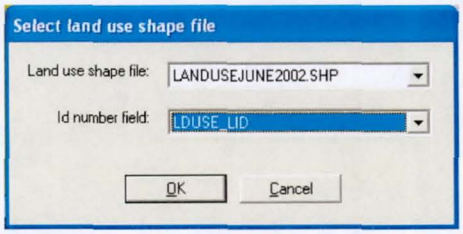
$$RTIMP = RTIMP_1 + \left(\frac{\sum A_i (RTIMPL)_i}{A_{GE}} \right)$$

Where:
Ai is obtained from land use shape file and
A_{GE} is the grid element area and *RTIMPL* is obtained from the 4th column of the land use table.

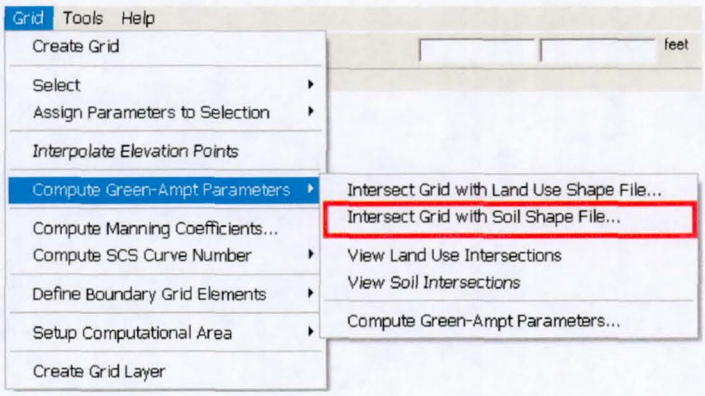
The steps to complete Green-Ampt parameter computation are as follows. Compute polygon intersections of the landuse shape file with each grid element.



Specify the landuse shape file and the ID number field as shown:

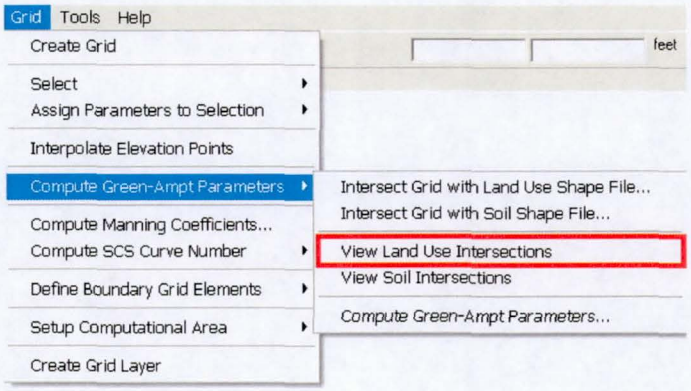


Click *'OK'* to compute. This procedure may take several minutes depending on the number of grid elements and the number of landuse polygons. Next compute the polygon intersections of the soil shape file with each grid element.

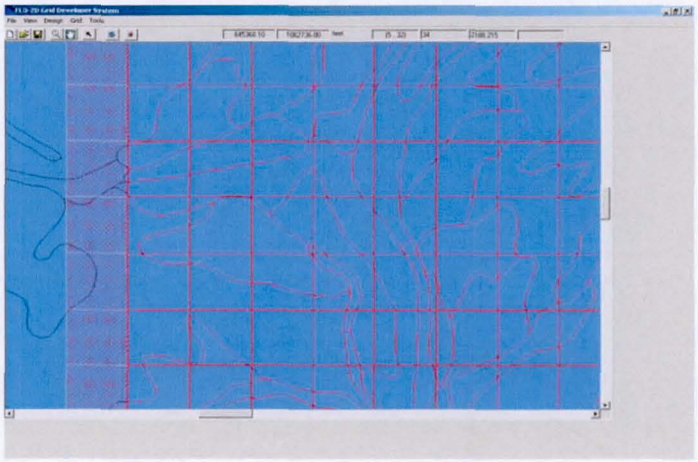


Again it is necessary to specify the soil shape file and the ID number field:

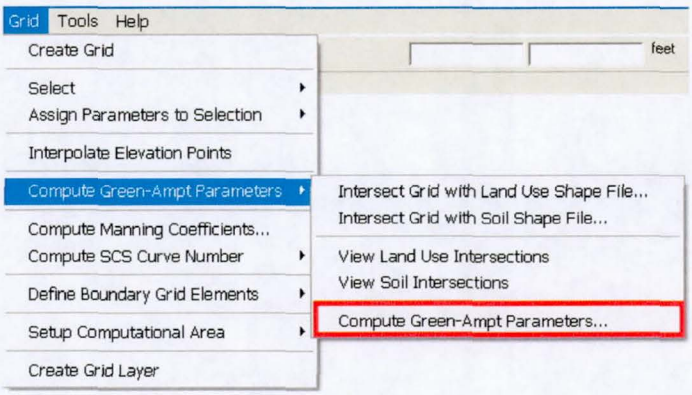
Click *'OK'* to compute. This procedure may take several minutes depending on the number of grid elements and soil type polygons. The computed intersected polygons can be viewed.



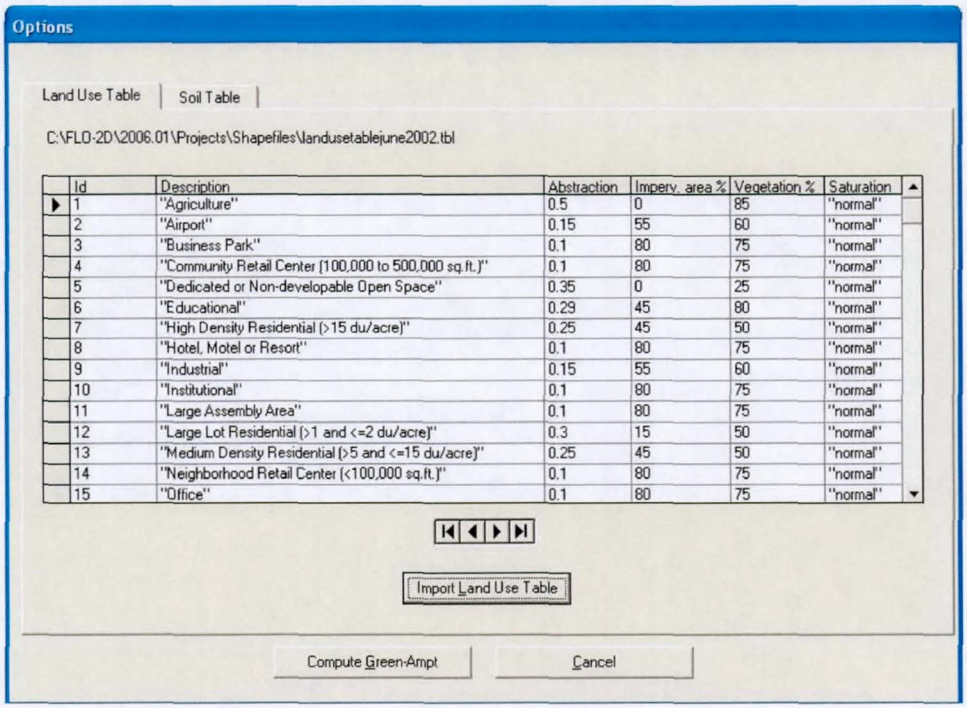
This is a zoom view showing soil polygons intersected with grid elements.



Now click on the *Compute Green-Ampt Parameters*.

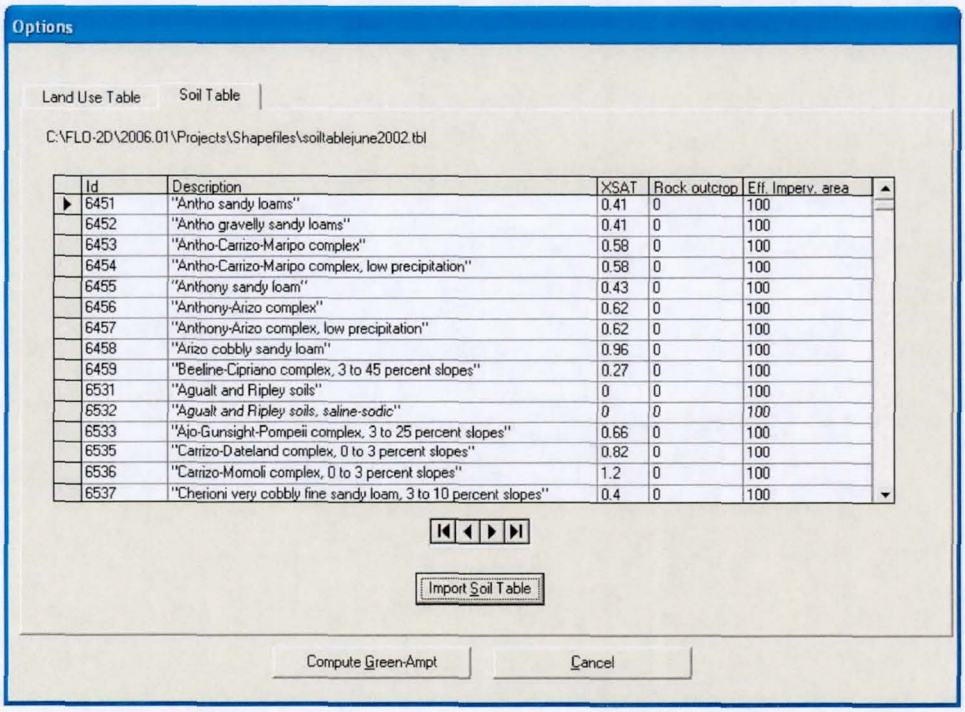


As part of this process, it is necessary to import the soil and landuse tables. First click on *Import Land Use Table* button and select the landuse table file to import. The data is displayed in the dialog box:



Id	Description	Abstraction	Impery. area %	Vegetation %	Saturation
1	"Agriculture"	0.5	0	85	"normal"
2	"Airport"	0.15	55	60	"normal"
3	"Business Park"	0.1	80	75	"normal"
4	"Community Retail Center (100,000 to 500,000 sq.ft.)"	0.1	80	75	"normal"
5	"Dedicated or Non-developable Open Space"	0.35	0	25	"normal"
6	"Educational"	0.29	45	80	"normal"
7	"High Density Residential (>15 du/acre)"	0.25	45	50	"normal"
8	"Hotel, Motel or Resort"	0.1	80	75	"normal"
9	"Industrial"	0.15	55	60	"normal"
10	"Institutional"	0.1	80	75	"normal"
11	"Large Assembly Area"	0.1	80	75	"normal"
12	"Large Lot Residential (>1 and <=2 du/acre)"	0.3	15	50	"normal"
13	"Medium Density Residential (>5 and <=15 du/acre)"	0.25	45	50	"normal"
14	"Neighborhood Retail Center (<100,000 sq.ft.)"	0.1	80	75	"normal"
15	"Office"	0.1	80	75	"normal"

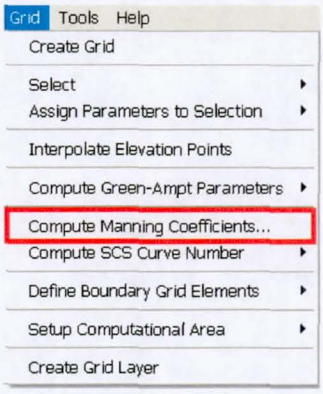
Now click on the *Soil Table* tab and click on the *Import Soil Table*. Select the appropriate soil table file. The data is displayed in the following dialog box:



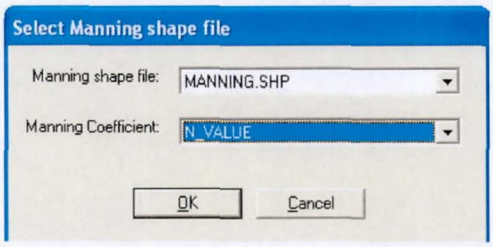
Id	Description	XSAT	Rock outcrop	Eff. Imperv. area
6451	"Antho sandy loams"	0.41	0	100
6452	"Antho gravelly sandy loams"	0.41	0	100
6453	"Antho-Carrizo-Maripo complex"	0.58	0	100
6454	"Antho-Carrizo-Maripo complex, low precipitation"	0.58	0	100
6455	"Anthony sandy loam"	0.43	0	100
6456	"Anthony-Arizo complex"	0.62	0	100
6457	"Anthony-Arizo complex, low precipitation"	0.62	0	100
6458	"Arizo cobbly sandy loam"	0.96	0	100
6459	"Beeline-Cipriano complex, 3 to 45 percent slopes"	0.27	0	100
6531	"Agualt and Ripley soils"	0	0	100
6532	"Agualt and Ripley soils, saline-sodic"	0	0	100
6533	"Ajo-Gunsight-Pompeii complex, 3 to 25 percent slopes"	0.66	0	100
6535	"Carrizo-Dateland complex, 0 to 3 percent slopes"	0.82	0	100
6536	"Carrizo-Momoli complex, 0 to 3 percent slopes"	1.2	0	100
6537	"Cherioni very cobbly fine sandy loam, 3 to 10 percent slopes"	0.4	0	100

The final step is to click on the *Compute Green-Ampt* button to create the INFIL.DAT file.

5.5.18 Compute Manning Coefficients… Command (Grid Menu)



This command will compute the Manning's n-values based on the information in a roughness shape file. With a project open, first import a Manning's n-value shape file using the *Import Shape File* command on the File menu. When you click on the *Compute Manning Coefficients* command, the following dialog box appears:



Select the Manning shape file name (MANNING.SHP in this case) and the Manning coefficient field (N_VALUE in this case) and click *'OK'* to calculate Manning coefficients for each grid element. Please note that the n-value data will be written to the FPLAIN.DAT only when you apply the *Create FPLAIN.DAT and CADPTS.DAT Files* command on the File menu or save the *FLO-2D* input files.

5.5.19 Compute SCS Curve Number ... Command (Grid Menu)

This command will compute the Curve Number for each grid element based on information contained in shape files.

The SCS runoff curve number loss method is a function of the total rainfall depth and the empirical curve number parameter which ranges from 1 to 100 and is a function of hydrologic soil type, land use and treatment, surface condition and antecedent moisture condition. The method was developed on 24 hour hydrograph data on mild slope eastern rural watersheds in the United States although runoff curve numbers have been calibrated or estimated for urban areas, agricultural lands and semi-arid range lands. The SCS CN method does not account for variation in rainfall intensity. The method was developed for predicting rainfall runoff from ungaged watersheds and its attractiveness lies in its simplicity. For large basins (especially semi-arid basins), the method tends to overpredict runoff using the standard curve number tables which has unique or variable infiltration characteristics such as channels (Ponce, 1989).

Assigning Curve Numbers to Grid Elements

The general procedure is to assign curve numbers in the *GDS* and then run the *FLO-2D* model with the infiltration component "turned on" and the infiltration switch set the SCS-CN option. The following options are available to assign curve numbers to grid elements:

1. Select area using a polygon and assigning a CN number. In this case the CN's are assumed known.
2. Import a CN polygon shape file and intercept the shape file polygons with the grid elements and interpolate the weighted average computed CN's. In this case the CN numbers are known.
3. Import three polygon shape files for hydrologic soil group (HSG), land cover, and impervious cover and the *GDS* will incept, interpolate and compute the CN's for each grid element based on the Pima County procedure. In this case the CN are not known

Determining CN's Using the Pima County Method

GDS uses the Pima County Method to determine the CN's using polygon shape files (SF) for soil, land cover and impervious cover. Using this shape file data with the accompany attribute tables,

the polygon intersection with the grid element are determined to assign a hydrologic soil group, percent cover density and impervious area for each grid element. Then the CN's are computed using various formulas depending on soil groups and the results are written to the *FLO-2D* INFIL.DAT file. The following procedure is applied:

1. Import soil, land cover and impervious cover shape files to the *GDS* graphical project environment.
2. Select the ID for the HSG from Soil shape file (landsoil.shp). The attribute may have both soil and group and the percentage for each group (e.g. B31%C69%Desert Brush). For each polygon the following attribute information is required:
    a. Cover type (e.g. Desert Brush)
    b. Soil group (e.g. B and C)
    c. Percentage of soil groups (B=35% and C = 69%). If only the group is given (e.g. B) then assume 100%.
3. Select ID for Cover density percentage from land cover shape file (land.shp).
4. Select ID for impervious areas from impervious cover shape file (imperv.shp)
5. Determine polygon intersections with each grid element for each SF (soil, land cover and impervious cover) and determine the area weighed average of each variable:

$$Var = \frac{\sum A_i Var_i}{A_{GE}}$$

Where:

Var_i is the current variable from the shape file;
A_i is subarea intercepted by the grid element and
A_{GE} the grid element area.

At the end of this process each grid element has been assigned:

    a. HSG from Soil shape file.
    b. Cover density percentage CD.
    c. Impervious area.
6. Use formulas in Table 1 to determine CN using CD and the corresponding soil group (e.g. CN = (CN soil B * (B%) + CN soil C * (C%) + %impervious* 99)/100) where: CN for impervious area is 99 and areas for each soil group within a grid element need to be adjusted depending on the impervious area: B% = B%(in SF) * (100-%impervious), etc.
7. Write CN results to the INFIL.DAT.

To determine CN's, functional relationships were derived from Figure D-1 of the Pima County Hydrology Procedures (Appendix D) reproduced here:

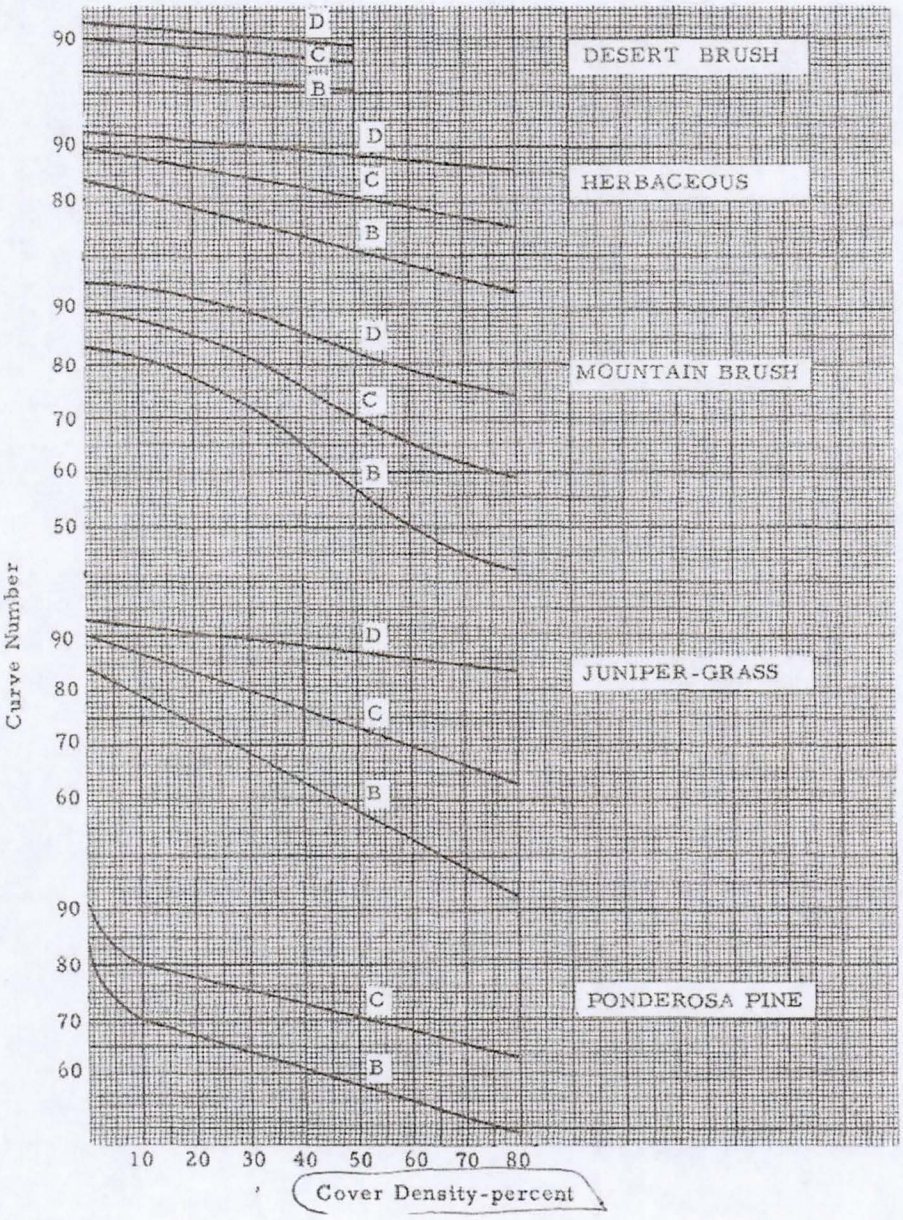


Figure D-1 of Pima County Hydrologic Procedures Relating CN to Hydrologic Soil Cover and Density

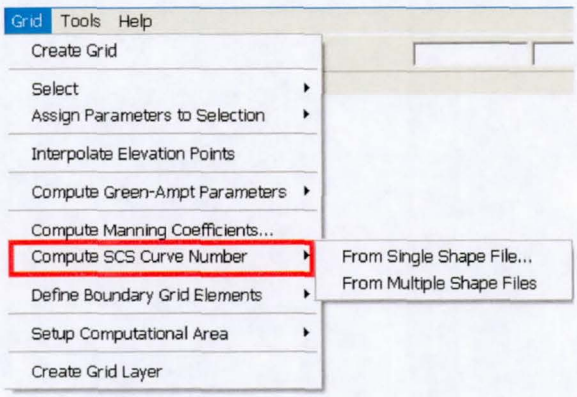
These relationships are summarized in the following table.

Table 1: Relationships for Evaluating the CN Based on Hydrologic Soil Cover and Cover Density

HSC		Cover density (CD) Range	Formula	A	B	C
DESERT BUSH						
	D	0-50	CN = A * CD + B	-0.08	93	
	C	0-50	CN = A * CD + B	-0.08	90	
	B	0-50	CN = A * CD + B	-0.07	84	
HERBACEOUS						
	D	0-80	CN = A * CD + B	-0.0875	93	
	C	0-80	CN = A * CD + B	-0.1875	90	
	B	0-80	CN = A * CD + B	-0.2625	84	
MONTAIN BRUSH						
	D	0-80	CN = A * CD^2 + B * CD + C	-0.0013	-0.1737	95
	C	0-80	CN = A * CD^2 + B * CD + C	-0.0014	-0.2942	90
	B	0-80	CN = A * CD^2 + B * CD + C	-0.0025	-0.3522	83
JUNIPER-GRASS						
	D	0-80	CN = A * CD + B	-0.1125	93	
	C	0-80	CN = A * CD + B	-0.34375	90.5	
	B	0-80	CN = A * CD + B	-0.525	84	
PONDEROSA PINE						
	C	0-10	CN = A * CD^2 + B * CD + C	0.08	-1.9	91
		10-80	CN = A * CD + B	-0.242857	82.42857	
	B	0-10	CN = A * CD^2 + B * CD + C	0.1	-2.4	84
		10-80	CN = A * CD + B	-0.3	73	
Legend						
HSC	Hydrologic Soil Cover					
CN	Curve Number					
CD	Cover Density (%)					
A, B	Constants					

For each soil group, a different formula was developed that calculates CN from the Cover Density (CD). For straight line segments, a linear equation was used to represent that soil cover. For curved segments of the , Excel curve fitting routines to determine the best-fit coefficients to represent the soil cover. Errors were estimated for each formula. Table 2 summarizes the validation and errors for each soil group set of formulas:

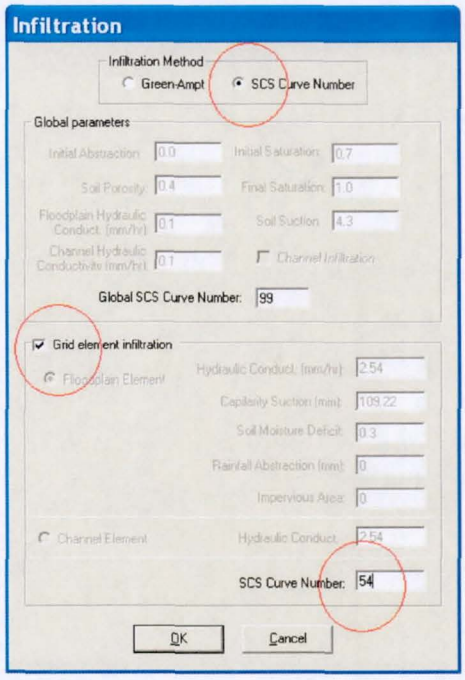
In the *GDS Grid* menu there is a new command titled: *Compute SCS Curve Number*. This has two options: From *Single Shape File...* and *From Multiple Shape Files*.



Assignment of CN's Based on Selection of Grid Elements

Using the existing procedures of Select/Assign Parameters to Selection, it is possible assign spatially varied CN to the grid elements. To use this feature, first select one or more grid elements by using the *Grid/Select/Grid Elements* and clicking on the elements that wish to be

selected. Use the *Grid/Assign Parameters to Selection/Infiltration* to display the infiltration dialog:

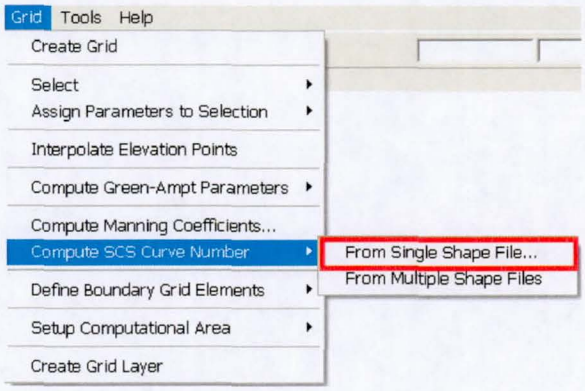


Select the SCS Curve Number radio button, select the Grid element infiltration check box and input the desired Curve Number. Click OK to assign the CN to the selected elements.

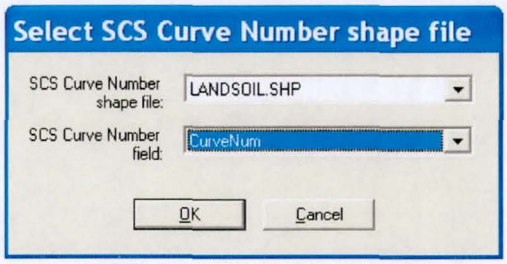
Assignment of CN's Based on a Single Shape File

Using this option, the *GDS* will determine the CN for each grid element based on a user provided polygon shape file that contains a CN for each polygon. First import the shape file using the *Import Shape File...* command in the *GDS File* menu.

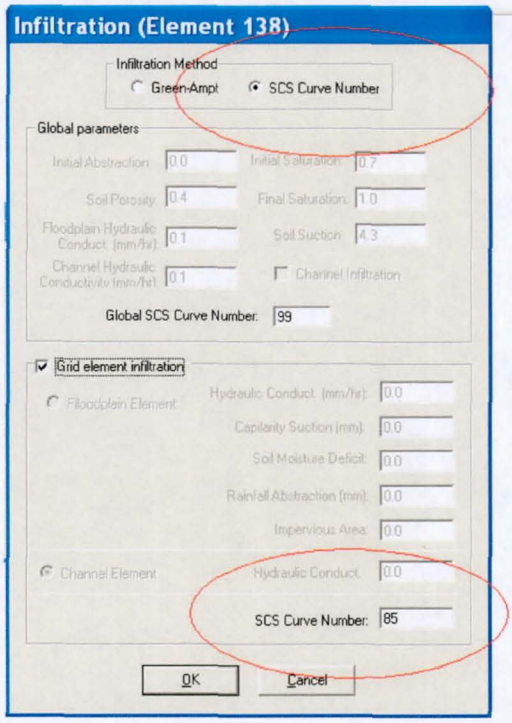
Then use the *Grid/Compute SCS Curve Number/From Single Shape File...* command



In the dialog box, select the Curve Number Shape file (LANDSOIL.SHP in this example) and the Curve Number Field (CurveNum in this example):

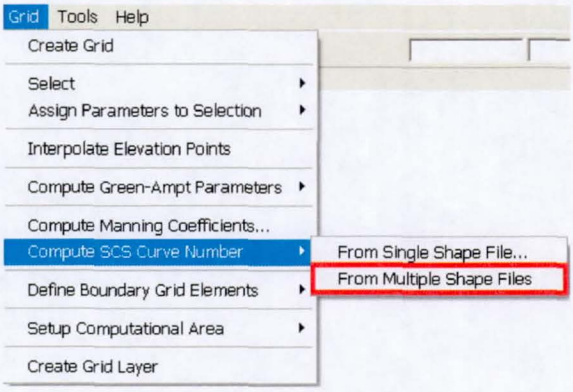


GDS will intersect the shape file polygons with the grid elements and determine the area weighted average value for each grid element. Double-clicking on any grid element the new Infiltration dialog box will display the CN calculated for that element.

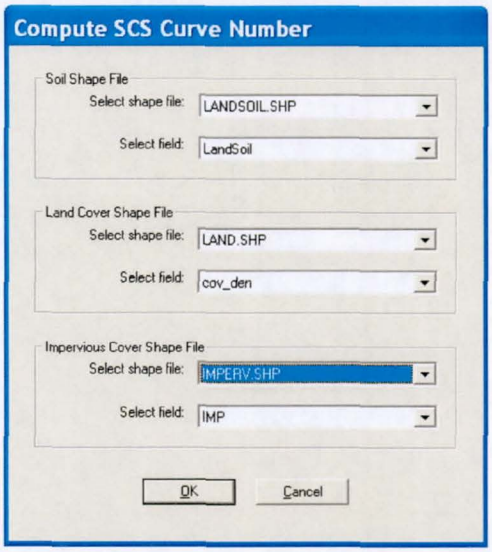


Computation of CN's Base on a Multiple Shape Files

This option computes the CN's based on the method explained in above sections. It is necessary to import three polygon shape files for HSG, Land cover, and Impervious cover. After these shape file have been imported using the *Import Shape File…* command in the *GDS File* menu, then use the *Grid/Compute SCS Curve Number/From Multiple Shape Files* command



The following dialog box will be displayed:

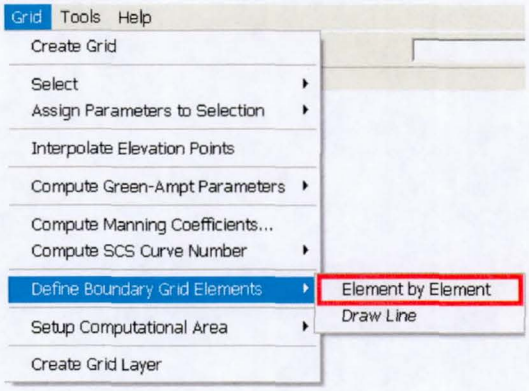


1. Select the Hydrologic Soil Group Shape File and soil group field (landsoil.shp and LandSoil respectively in this example).
2. Select the Land Cover Shape File and land cover attribute field (land.shp and cov_dens respectively in this example).
3. Select the Impervious Area Shape File and impervious ID filed (Imperv.shp and IMP respectively in this example).
4. Click OK.

GDS will intersect the shape files polygons with the grid elements and determine the area weighted average value for each grid element, based on the derived formulas previously discussed. Double-clicking on any grid element the new Infiltration dialog box will display the CN calculated for that element. *GDS* will create the new INFIL.DAT data file with the new CN's for all grid elements.

FLO-2D Infiltration Computation Using the SCS Curve Number Method *GDS* will create the new INFIL.DAT data file with the CN for all grid elements.

5.5.20 Define Boundary Grid Elements/Element by Element Command (Grid Menu)



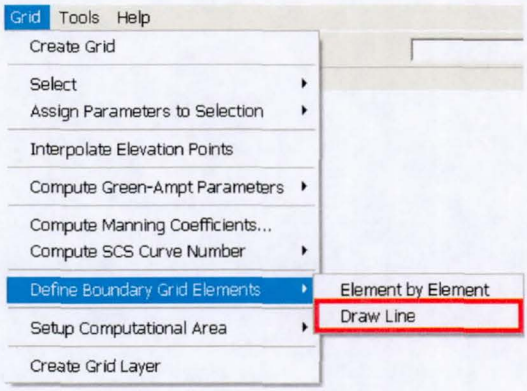
Use this command or the Define Boundary Grid Elements toolbar icon to mark the grid elements that will constitute the computational domain boundary. These boundary grid elements are not used in a *FLO-2D* model flood simulation and are not included in either the FPLAIN.DAT or CADPTS.DAT files. To use the *Define Boundary Grid Elements* command or icon:

1. Select the *Define Boundary Grid Elements (Element by Element)* command (Grid menu) or the toolbar icon and click on the left mouse button. The mouse pointer changes to a cross.
2. Select the boundary grid elements. The grid element will change its color to red.
3. Repeat step 2 as many times as needed to select all boundary grid elements.

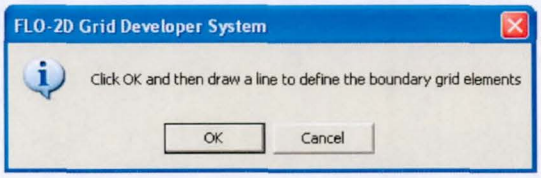
To undo this operation simply click again on a previously marked grid element. To mark a group of grid elements press the **Shift** key and the left mouse button and then drag the mouse pointer over the desired grid elements. To deselect any group of grid elements, press the **Ctrl** key and with the left mouse button depressed, drag the mouse pointer over the grid elements you want to unmark.

5.5.21 Define Boundary Grid Elements/ Draw Line Command (Grid Menu)

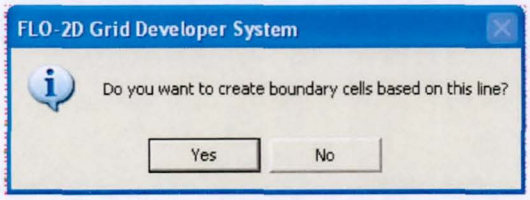
The boundary grid elements can also be identified by drawing a line around the computation domain. This command automates the process of defining the boundary elements. After the boundary grid elements have been marked with the line, they can be edited with the *Element by Element* command or the toolbar icon.



Select this command after the grid system has been generated. When you click on *Draw Line*, the following dialog box appears:

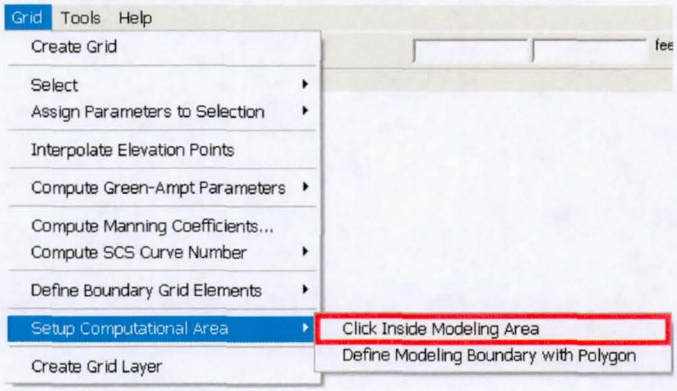


Click *'OK'* and proceed to draw a polygon with a series of left mouse button clicks. Complete the polygon by doubling clicking on the final vertex and then click *'Yes'* in the following dialog box to generate the boundary elements.



If the boundary elements identified by the polygon are acceptable, then generate the computation domain based on enclosed polygon by clicking *'Yes'* when requested to do so.

5.5.22 Setup Computational Area/Click Inside Modeling Area Domain Command (Grid Menu)



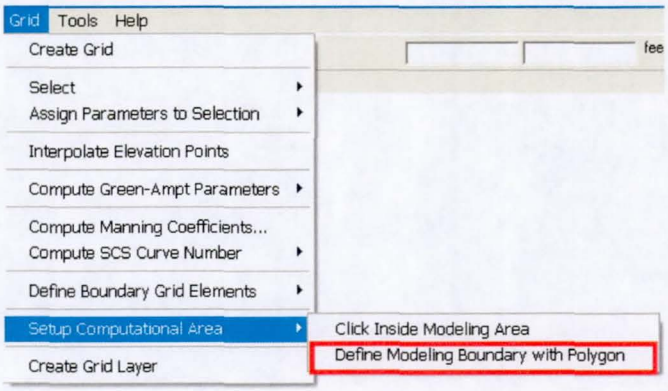
Use this command to select the computational domain. Once you have finished outlining the flow domain with a closed boundary, you must identify computational domain (potential flow surface).

1. Choose the *Setup Computational Area* command. The cursor changes to a cross.
2. Click on any grid element inside the closed area of the computational domain. If the computational domain area is closed, an acceptance message appears. If the boundary elements do not completely enclose an area, you will be informed that the area is still open. Additional boundary grid elements must then be marked to completely close the computational domain.

If you have defined the boundary cells element by element, or if you have edited the boundary cells, the computational domain can be generated by clicking on an interior grid element within the domain using the *Click Inside Modeling Area* command.

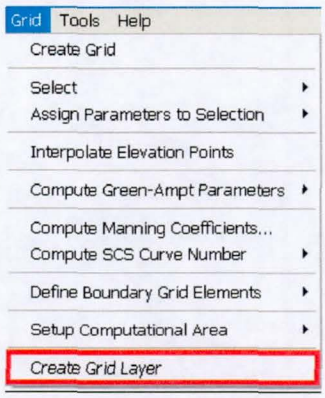
5.5.23 Setup Computational Area/ Define Modeling Boundary with Polygon Command (Grid Menu)

As was previously discussed, you can define the computational domain by drawing a polygon to identify the boundary cells.



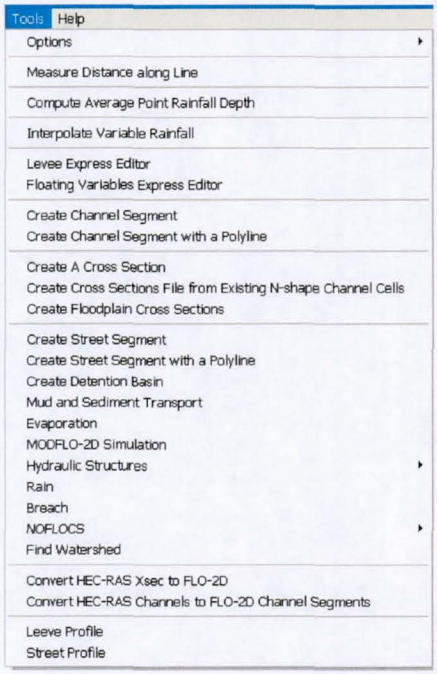
Follow the procedure described in Section 3.6 to define the computation domain with a polygon.

5.5.24 Create Grid Layer Command (Grid Menu)

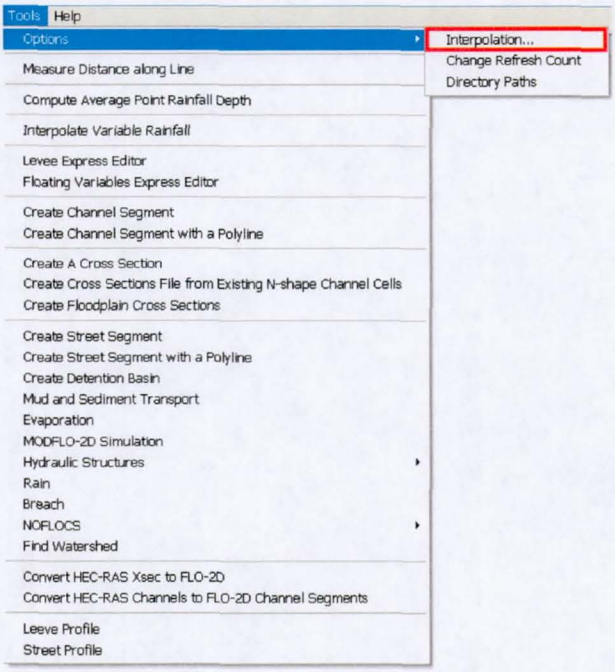


This command will create the grid layer. When loading an existing *FLO-2D* project the grid layer is not created. Some component editor functions require the grid layer and a message will you prompt you to create this layer. Use this command to create the grid layer and continue with grid element attribute editing.

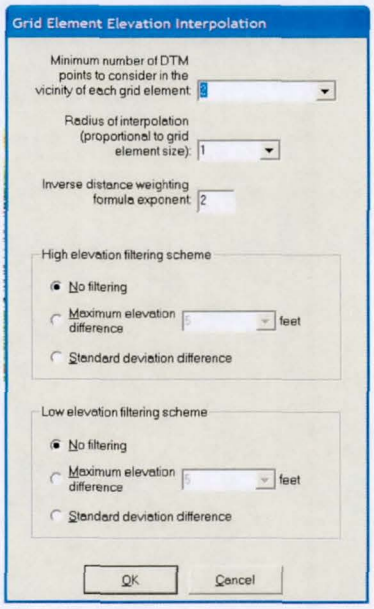
5.6 **Tools Menu Commands:**



5.6.1 Options…/Interpolation… Command (Tools Menu)



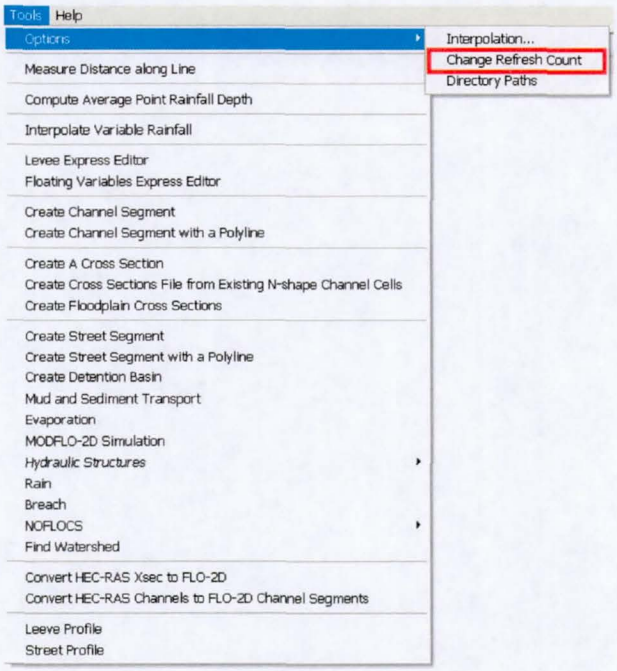
This command defines the various options that control the interpolation process. When command is selected, a dialog box appears to adjust the various interpolation parameters:



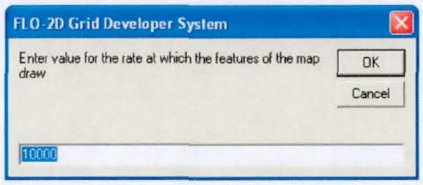
Dialog Box Options:

Option	Description
Minimum number of DTM points to consider in the vicinity of each grid element	To calculate an interpolated elevation from random DTM points that will represent each grid element, the algorithm will use at least this minimum number of DTM points closest to the grid element center.
Radius of interpolation (proportional to grid element size)	The radius defines a circle around each grid element node that is multiple of the element size. Only the DTM points inside the radius are interpolated. If an insufficient number of elevation points are within the circle, it is expanded until at least the minimum number of points is found.
Inverse distance weighting formula exponent	This is "n" in the Inverse Distance Weighting Formula: $\bar{Z} = \frac{\sum_{j=1}^{NDTM} Z_j \frac{1}{r_{ij}^n}}{\sum_{j=1}^{NDTM} \frac{1}{r_{ij}^n}}$ Where $\bar{Z}$ is the interpolated grid element elevation, Z_j is the elevation of DTM point $_j$, r_{ij} is the distance from the DTM point $_j$ to center of grid element $_i$ and NDTM is the total number of DTM points.
No filtering (High or Low Elevations)	No filtering is performed on the elevation points.
Maximum elevation difference (For either High or Low Elevations)	For this option, the algorithm calculates a mean elevation using all the DTM and any assigned elevation points within the interpolation radius. All those points that exceed the selected maximum elevation difference either higher or lower than the mean are discarded and the mean elevation is recomputed and assigned to the grid element.
Standard deviation difference (For Either High or Low Elevations)	The algorithm first calculates the DTM point elevation standard deviation and then neglects all those points whose elevations are higher (or lower) than the standard deviation when recalculating grid element mean elevation.

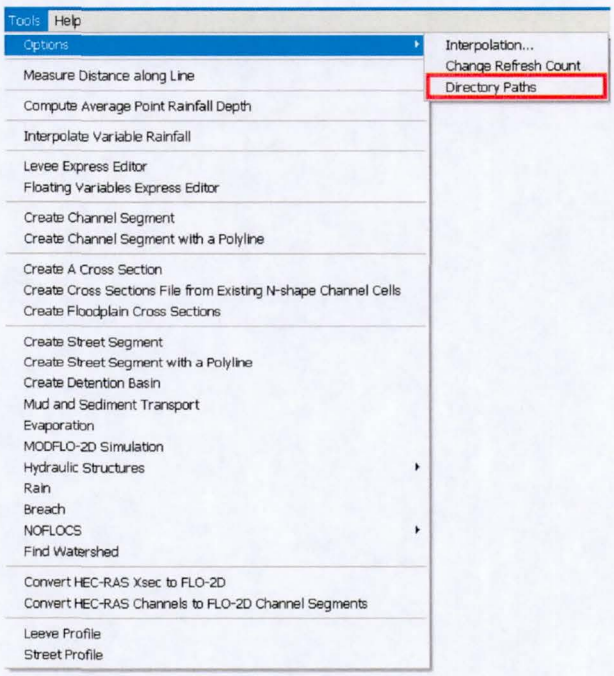
5.6.2 Options…/Change Refresh Count Command (Tools Menu)



Use this command to speed up the redraw (refresh) rate for large projects. The refresh rate specifies how often the screen is updated when drawing maps. If there are 1000 vertices to draw and the refresh rate is 500, then the screen will be refreshed twice. A higher refresh rate results in faster draw times while a lower rate results a smoother draw appearance. A dialog box will appear prompting you to input a value for the refresh rate of the map or image:

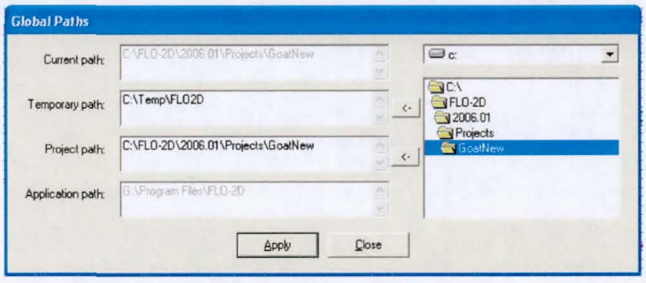


5.6.3 Options…/Directory Paths Command (Tools Menu)



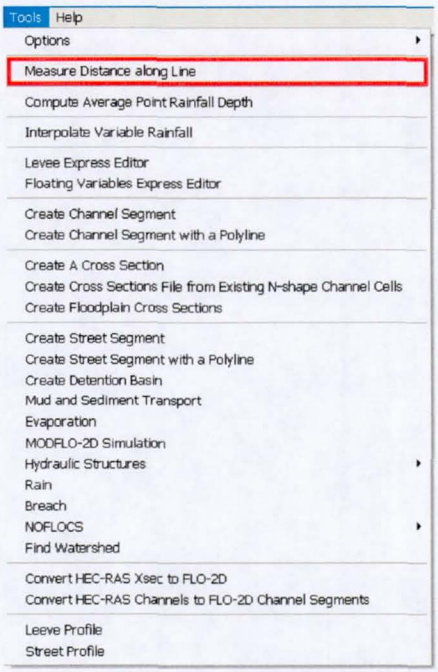
V

This command may be used to edit the default directories used by the *FLO-2D* model. The following dialog box appears:



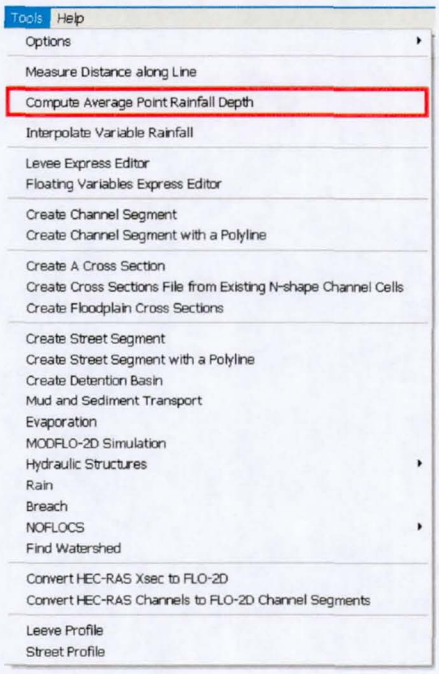
Input the desired new directory paths and then click on *Apply*.

5.6.4 Measure Distance along Line Command (Tools Menu)

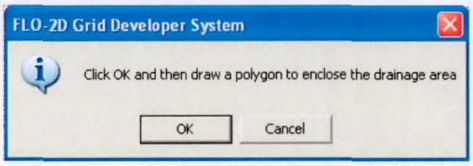


This tool computes the distance along a user-input polyline. After clicking on the *Measure Distance along Line Command,* draw a polyline using the mouse and double click at the end point of the polyline. *GDS* will display the length of the polyline as shown bellow:

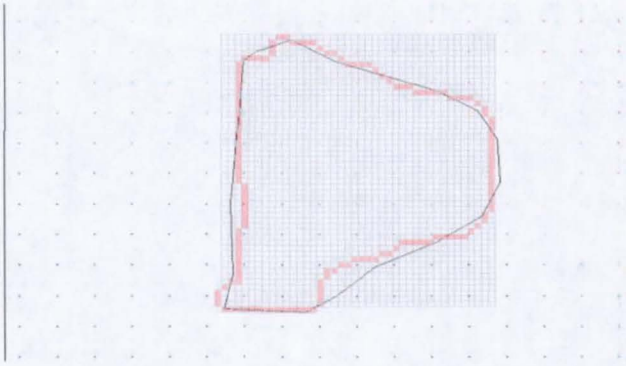
5.6.5 Compute Average Point Rainfall Depth Command (Tools Menu)



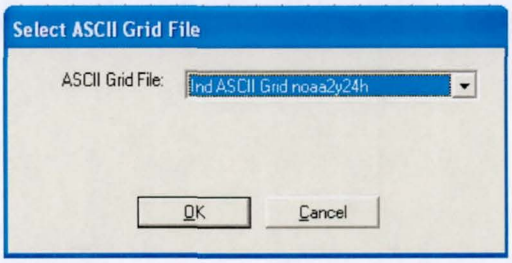
Use this command to compute Total Point Rainfall Depth (RTT in RAIN.DAT file). *Note: It is required that a RAIN.DAT file exist in the project folder.* An ArcInfo ASCII grid file with rainfall data must be previously imported to the *GDS*. The format of this file is the standard ASCII grid with the values corresponding to rainfall depths. The points will be displayed on the screen. To compute the average of the grid rainfall depth, draw a polygon and then only the rainfall points inside the polygon will be used in the calculation.



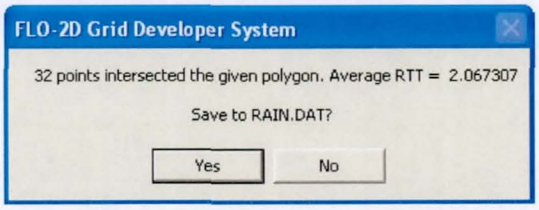
The following figure displays a *FLO-2D* grid system boundary, the ASCII rainfall point depth grid and the drawn polygon to compute that average rainfall depth over the *FLO-2D* grid system.



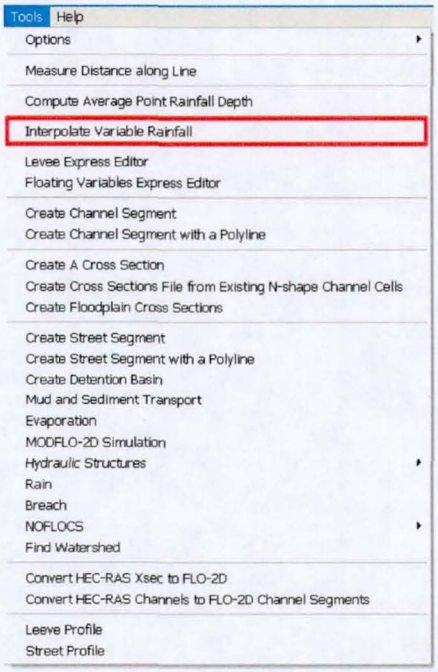
To compute the average rainfall depth, choose the appropriate ArcInfo grid file from the file list in the following dialog box:



The following message will identify the number of grid points used to compute the average rainfall depth RTT value. The RAIN.DAT file RTT value will be updated.



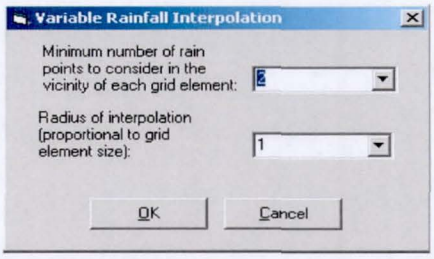
5.6.6 Interpolate Variable Rainfall Depth Command (Tools Menu)



This command will interpolate temporal rainfall for each *FLO-2D* grid element, based on spatially varied rainfall data in an ArcInfo ASCII grid file format. A file must be prepared that contains the storm beginning and ending dates and times, time interval of the rainfall data files (in minutes), number of reporting storm time intervals (one rainfall data file per interval) and the file name and location. Each subsequent rainfall data file will list the rainfall depth at a rain gage grid location (i.e. rainfall grid not the *FLO-2D* grid system) at that specific time. For example:

6/12/2002 13:00 6/12/2002 14:00 60 2
C:\Models\FLO2D_2002\Maricopa2002\Raindata2\rain13h.dat
C:\Models\FLO2D_2002\Maricopa2002\Raindata2\rain14h.dat

In this case, the beginning date of the storm is 6/12/2002 and the storm beginning time is 13:00 hours. The storm ending date and time is 6/12/2002 and 14:00 hours. The time interval between rainfall data files is 60 minutes and there are two rain grid files in the list. The rainfall file format is the standard ArcInfo ASCII grid with the values corresponding to rainfall depths. Each file corresponds to a rainfall depth at a specific time for the rainfall grid system. The rainfall grid file (e.g. NEXRAD data) is prepared outside the *FLO-2D* software system. This file may be based on the rainfall gage distribution. This rainfall distribution over the rainfall grid has to be transformed into spatially varied rainfall assigned to grid elements in the *FLO-2D* grid system. This is accomplished using the variable rainfall interpolation algorithm in the *GDS* program. Once the ArcInfo ASCII file is read, the following dialog box permits adjustment of rainfall grid interpolation parameters.



Option	Description
Minimum number of rain grid points to consider in the vicinity of each grid element.	To calculate the interpolated rainfall depth for each *FLO-2D* grid element, the algorithm uses at least this minimum number of points closest to the grid element.
Radius of interpolation (proportional to grid element size).	This radius defines a circle around each grid element based on a multiple of the grid elements size. Only rainfall grid points inside this radius are considered in the interpolation process. If the number of rainfall grid points is insufficient, then the circle is automatically enlarged until at least the minimum number of points is found.

After the interpolation is complete, choose the RAINCELL.DAT file location to save results. The RAINCELL.DAT file is then used by the *FLO-2D* model to simulate the rainstorm. The IREALRAIN variable switch in the *FLO-2D* RAIN.DAT file has to be set to 1 'on' to simulate a real-time storm using the RAINCELL.DAT file.

The format of the RAINCELL.DAT is as follows:

Line 1: Beginning date and time, ending date and time, time interval (in minutes) and number of time intervals.

Line 2: Grid element number and the total rainfall depth at that time. This line is repeated for each *FLO-2D* grid element.

Successive rainfall data for the entire grid system follows for the number of time-interval groups. For the example above, there would be two sets of rainfall data for the entire grid system.

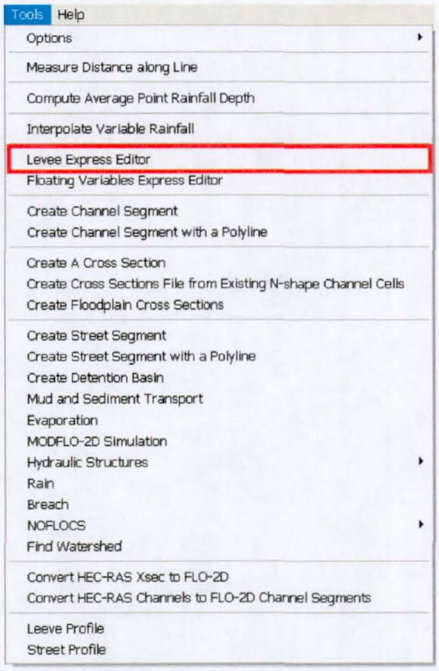
6/12/2002 1:00:00 PM 6/12/2002 2:00:00 PM 60 2

```
1      1.8538
2      1.8537
.        .
1234   2.272
1235   2.2709

1      1.4794
2      1.4791
.        .
1234   1.6294
1235   1.6286
```

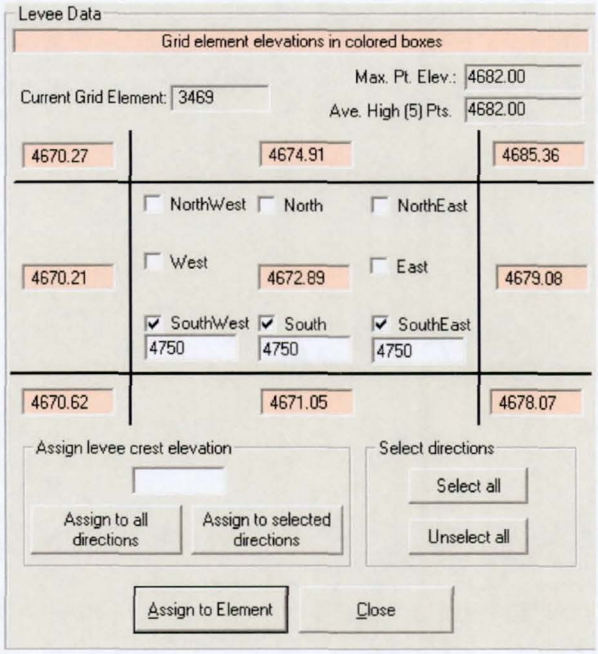
In this above example the beginning date is 6/12/2002 the beginning time 1:00 PM, the ending date 6/12/2002 and the ending time 2:00 PM. The time interval between files is 60 minutes, there are two time intervals and 1235 grid elements. The rainfall depth for grid element number 1 at time 1:00 PM is 1.8538 in. Successive grid element rainfall depths follow. For the second time interval, the rainfall depth for the first grid element is 1.4794 in and the rest of the grid elements follow. If the grid system has been prepared in International Units (meters), then the rainfall data file should be prepared in mm.

5.6.7 Levee Express Editor Command (Tools Menu)



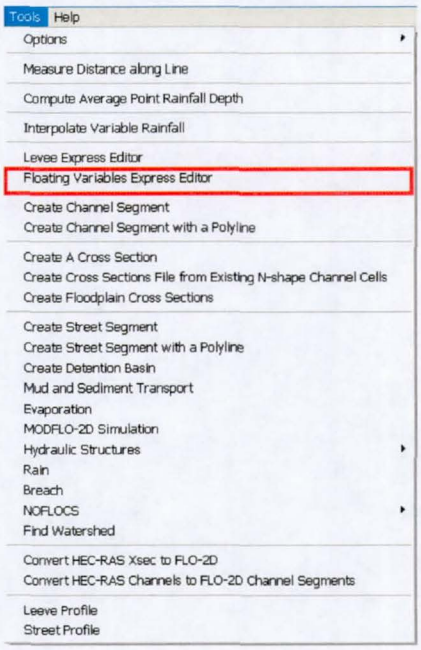
The Levee Express Editor facilitates assigning levee attributes for individual grid elements. You can use the Levee Express Editor to create or edit a levee. This tool will help you to assign levee crest elevations directly from the DTM point data base, if the DTM points are dense enough to represent the levee crest. If a number of DTM points fall on the top of the levee, the average of the five highest DTM points can be used to represent the levee crest.

When you click on this command or on the toolbar icon for an existing levee the following floating dialog appears:

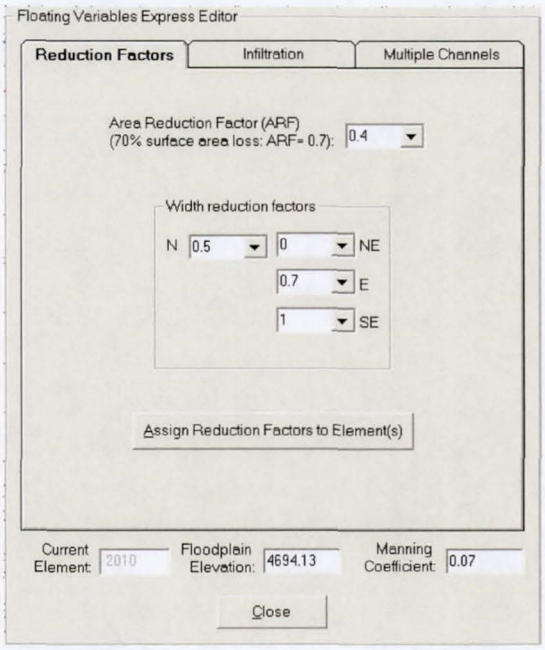


When you double click on a grid element that has a levee, the dialog will show the Maximum Point elevation of the DTM points inside the element, the average of the highest 5 DTM points in the element and the surrounding grid element elevations. Use the text boxes and the *Assign to Element* button to assign the levee crest elevations. Compare the Maximum Point Elevation (Max. Pt. Elev.) and the Average of the Five Highest Points (Ave. High (5) Pts.) with the upstream and downstream assigned levee crest elevations to assign the levee crest elevation for the given grid element.

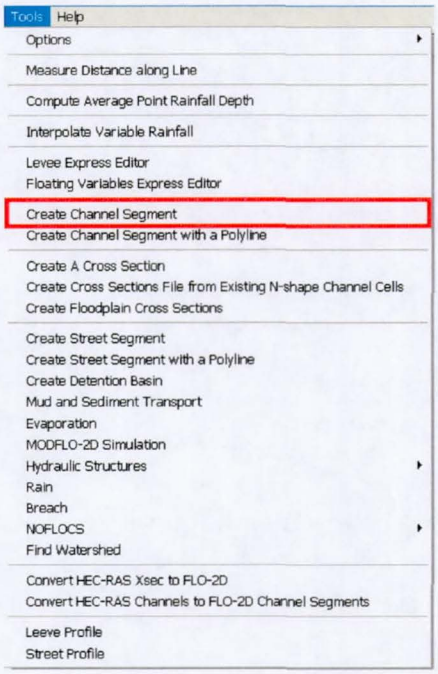
5.6.8 Floating Points Variables Express Editor Command (Tools Menu)



This command opens the Floating Variables Express Editor that allows interactive editing of Area and Width Reduction Factors (ARF,WRF), Infiltration parameters and Multiple Channel Data. While the dialog remains open and you double click on an element the data corresponding to the active tab will be loaded, where it can be edited.

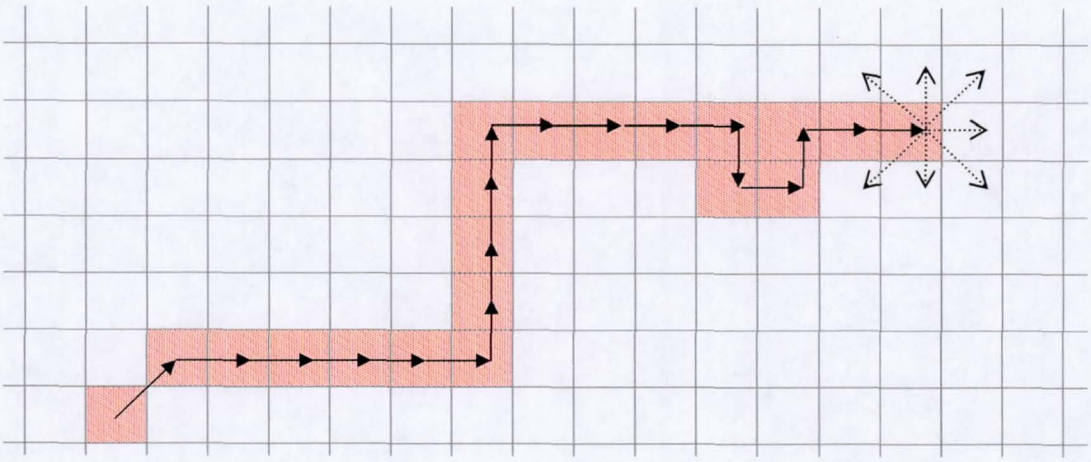


5.6.9 Create Channel Segment Command (Tools Menu)



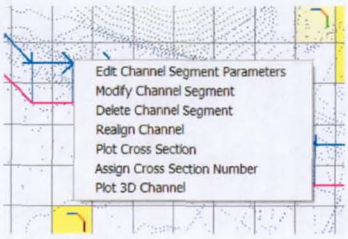
This command creates the channel network in an existing *FLO-2D* grid system using point and click tools as described below:

1. On the *Tool menu* click *Create Channel Segment*
2. The mouse pointer changes to a cross.
3. To add a new grid element to the channel segment, click on first (upstream) grid element of the segment. While holding the left mouse button down, select channel elements as you move the mouse over the grid system. Use a background image to locate the channel with respect to the grid system. Channel elements are joined by arrows to visualize the channel location:

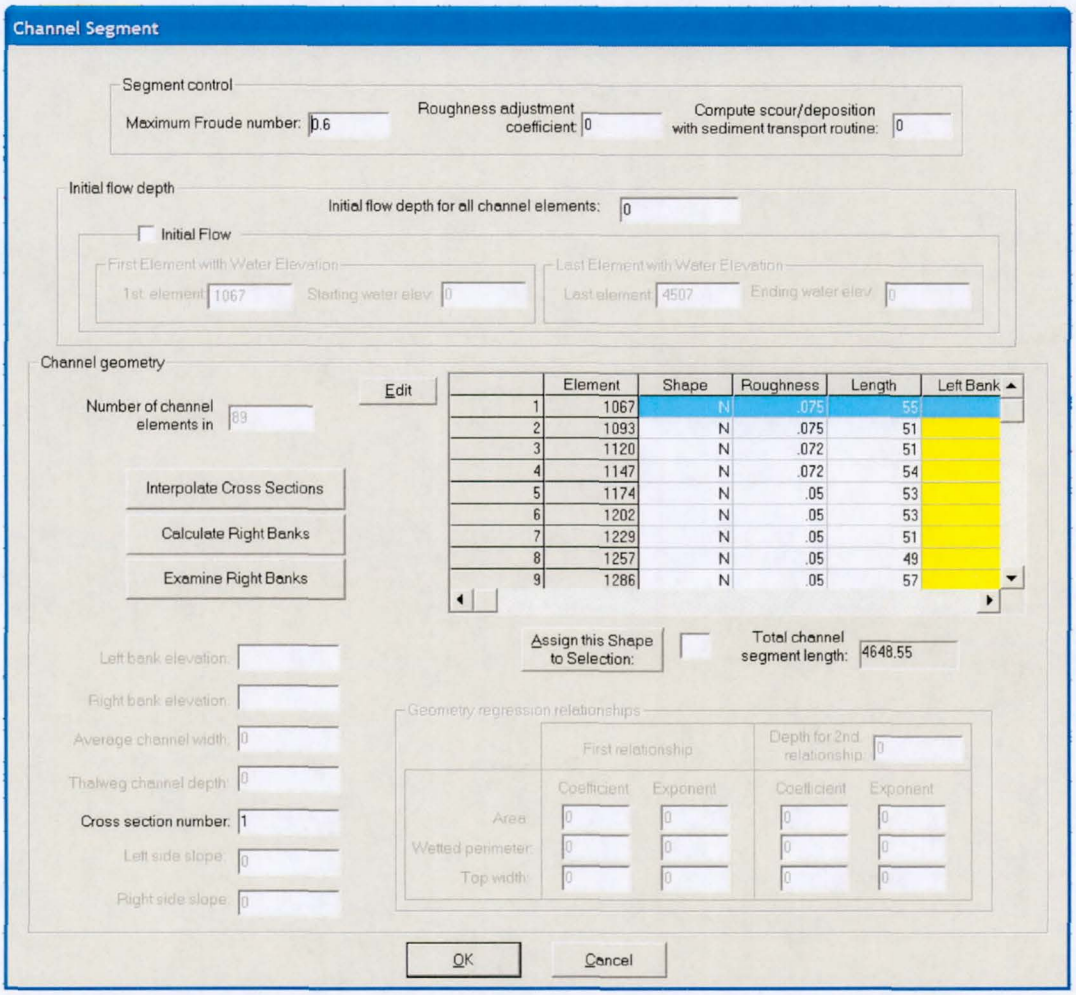


As the channel development progresses from one element to the next, only the eight adjacent elements can be accessed (see above figure). When the mouse pointer is near the center of the next element to be selected, a solid arrow appears to indicate which element will be the next one in the segment. The user can backup the pointer over the segment to unselect channel grid elements. No clicks are necessary, the users only needs to sweep the pointer over the desired elements. The channel selection is complete when the user double-clicks with the left mouse button.

To edit channel geometry parameters, click on a channel segment. A submenu appears with several options:



Select *Edit Channel Segment Parameters* and to display the Channel Segment dialog box:



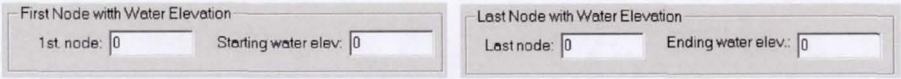
The channel data is governed by the following rules (refer to the CHAN.DAT file description in the *FLO-2D* Data Input Manual for more detailed discussion):

1. *Maximum Froude Number*: Set the channel roughness to a reasonable n-value and then set the FROUDC variable to an appropriate value (e.g. 0.95 to ensure subcritical flow). *FLO-2D* will adjust the roughness values according to the limiting Froude number criteria.

2. *Roughness adjustment coefficient*: This coefficient is used for depth variable Manning's n-values. This value can range from 0.00 to 0.4.

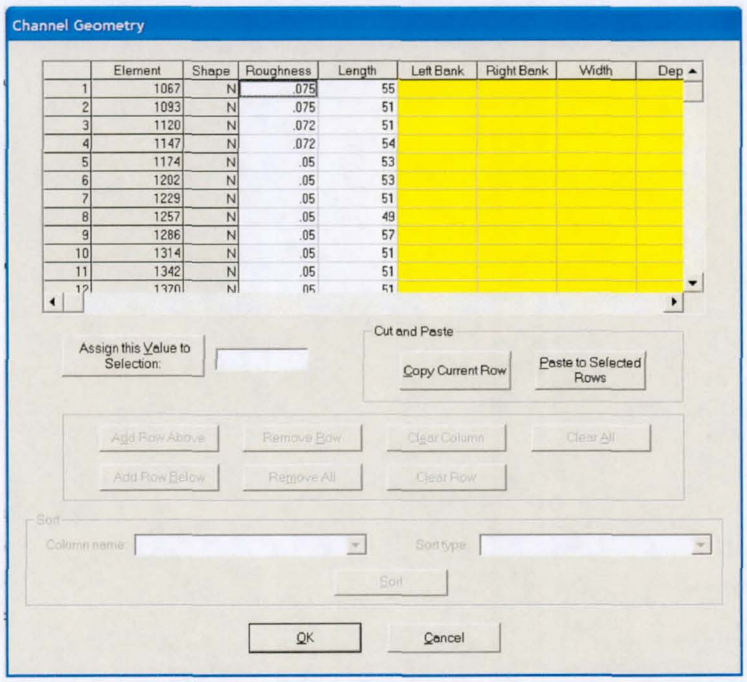
3. *Compute scour/deposition with sediment transport routine*: This is a check box to compute sediment transport in the channel segment. Checked the box if sediment transport is to be computed. In addition, ISED must be turned "On" in CONT.DAT.

4. *Initial flow depth*: When the *initial flow depth for all channel nodes* is > 0, an initial depth is specified for all the channel elements in that segment. Checking the *Initial Flow* check box ☑ Initial Flow requires assigning starting and ending water surface elevations for the channel segment beginning with element *1st node* and ending with element *last node.*



Interpolated water surface elevations are assigned to the channel elements in that segment based on the channel length.

5. Each row in the "Channel Geometry" table corresponds to a channel grid element.
6. The "Geometry Regression Relationships" data table is enabled only when a variable geometry channel element of type "V" is selected in the "Channel Geometry" table.
7. Depending on the channel element data base some columns may be disabled.
8. The editing of channel element data in each segment is achieved by pressing the "Edit" button. Interact with the table in the following dialog window to edit the channel element data (node, extension direction, roughness and length):



	Element	Shape	Roughness	Length	Left Bank	Right Bank	Width	Dep
1	1067	N	.075	55				
2	1093	N	.075	51				
3	1120	N	.072	51				
4	1147	N	.072	54				
5	1174	N	.05	53				
6	1202	N	.05	53				
7	1229	N	.05	51				
8	1257	N	.05	49				
9	1286	N	.05	57				
10	1314	N	.05	51				
11	1342	N	.05	51				
12	1370	N	.05	51				

Additional channel data instruction comments can be found in the CHAN.DAT file descriptions. A few comments are highlighted here:

- Dividing the channel into segments may simplify reviewing the results. Organize the channel segments and elements from upstream to downstream with the inflow grid element being the first grid element in the file or segment.
- The key to channel routing is to balance the relationship between the slope, flow area and roughness. Channel routing is more stable if surveyed cross section data is used. Review the PROFILES program instructions for cross section and channel bed slope interpolation.
- If channel geometry is being simulated with regression relationships (SHAPE = 'V'), then the area (A) versus depth (d) power relationships ($A = ad^b$) coefficients (a) and exponents (b) must be specified. Similar relationships are required for wetted perimeter and top width. If two power relationships are used to represent a natural cross section,

then the maximum depth (*Depth for 2nd relationship variable*) to which the first relationship applies must be specified. Some rules for applying the second variable area channel geometry relationships follow:

i. The second regression applies when the flow depth is greater than *Depth for 2nd relationship variable*, but does not include the lower flow area. The two variable area cross section relationships are unique and separate. The total cross section flow area is the sum of the lower flow and upper (second relationship) flow areas. The channel top width is computed directly from the second relationship. The area, wetted perimeter and top width are evaluated using the upper flow depth given by: (total depth - *Depth for 2nd relationship variable*). To analyze the upper channel geometry using the XSEC program, only the cross section coordinates above the *Depth for 2nd relationship variable* are used.

ii. Channel geometry relationships apply only to flow depths that are less than the channel depth (lower than the top of bank). When the flow depth exceeds the top of bank, then the channel geometry above bank is evaluated as a rectangle.

iii. Abrupt transitions between contiguous channel elements should be avoided unless they actually exist.

iv. A preprocessor program XSEC is available in the *FLO-2D* subdirectory to determine the regression coefficient and exponents.

- Channel elements that are contiguous but do not share discharge (e.g. parallel channels) must be identified with the NOFLO1 and NOFLO2 variables. List each pair of contiguous channel elements only once.
- Channel elements that will not share discharge will the floodplain have to be identified by the NOEXCHANGE parameter.
- To improve the timing of the floodwave progress through the system, a depth variable roughness can be assigned on a reach basis. The basic equation for the channel element roughness n_d as function of flow depth is:

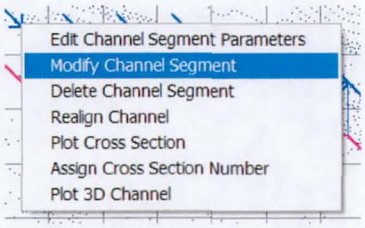
$$n_d = n_b \, r_c \, e^{-(r2 \text{ depth/dmax})}$$

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = bankfull flow depth
r2 = roughness adjustment coefficient prescribed by the user (range: 0. to 0.4)
$r_c = 1./e^{-r2}$

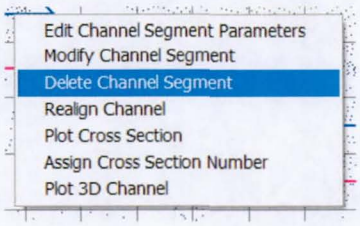
This equation prescribes that the variable depth channel roughness is equal to the bankfull roughness at bankfull discharge. If the user assigns a roughness adjustment coefficient r2 for a given reach, the roughness will increase with a decrease in flow depth. The higher the coefficient, the greater the increase in roughness.

To modify the channel elements in the channel segment, select the *Modify Channel Segment* Command:

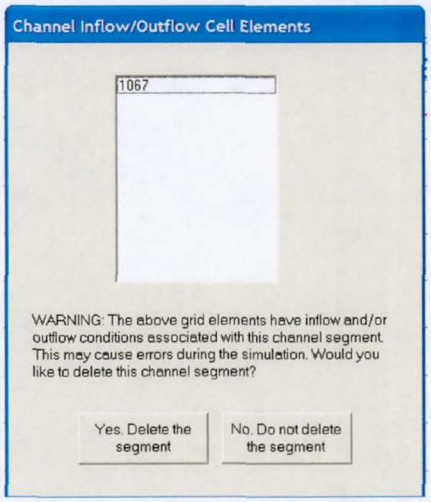


You have the same interactive graphic tools to add or delete channel elements as described above in the *Create Channel Segments* Command.

To delete an entire selected channel segment, select the *Delete Channel Segment* command:

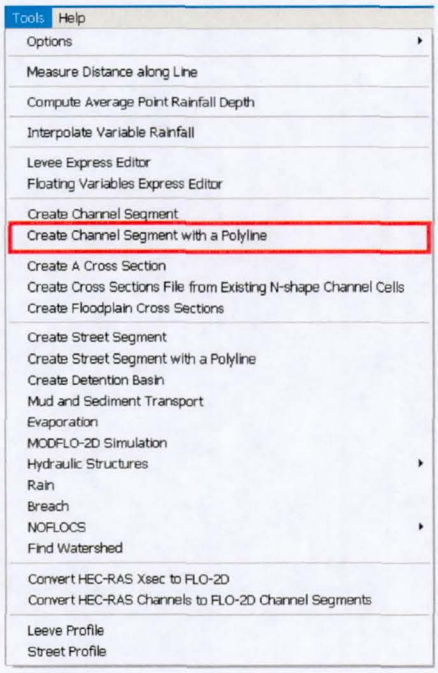


If the channel segment has assigned inflow hydrographs the following warning will appear:



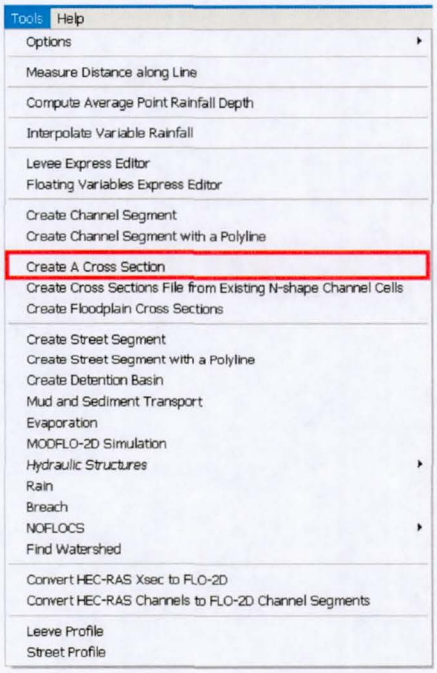
If you select Yes, you will need to click on the inflow node(s) that was assigned to the segment and reassigned to the floodplain instead.

5.6.10 Create Channel Segment with a Polyline Command (Tools Menu)

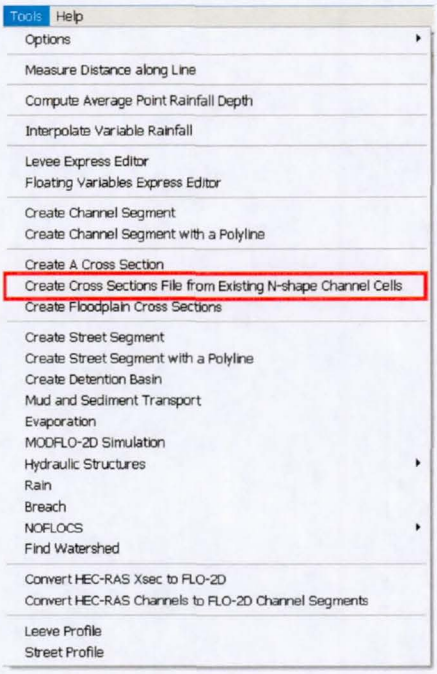


With this command you can create a channel segment more efficiently using simple mouse point and clicks. First click on *Create Channel Segment with a Polyline* command; the cursor will change to a cross and as you click the mouse, the channel segment will be progressively created. To complete the segment, just double-click.

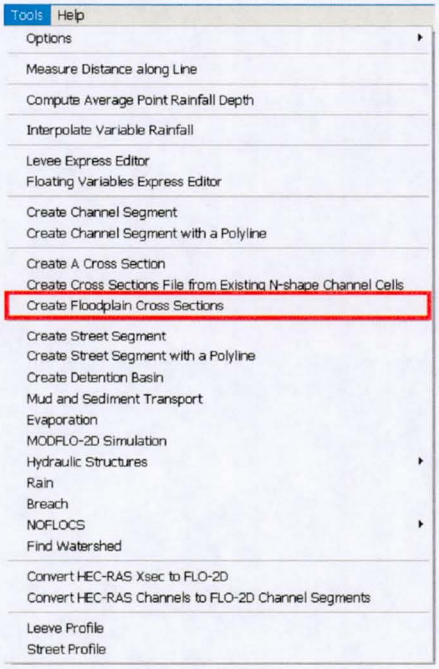
5.6.11 Create Cross Section Command (Tools Menu)



5.6.12 Create Cross Section File from Existing N-shape Channel Cells Command (Tools Menu)

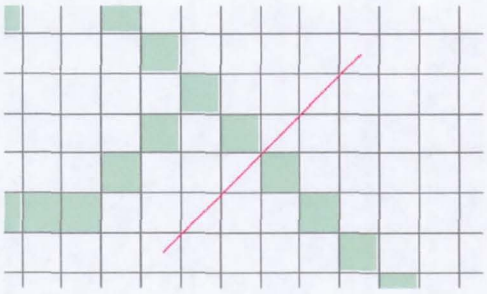


5.6.13 Create Floodplain Cross Sections Command (Tools Menu)

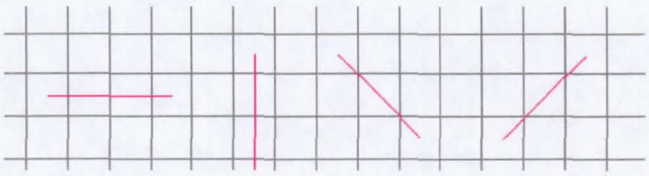


This command will enable selection of grid elements for a floodplain cross section so that the *FLO-2D* model will calculate a hydrograph and compile hydraulic results for the flow across the cross section. You can create the cross section in an existing *FLO-2D* grid using point and click tools as described below:

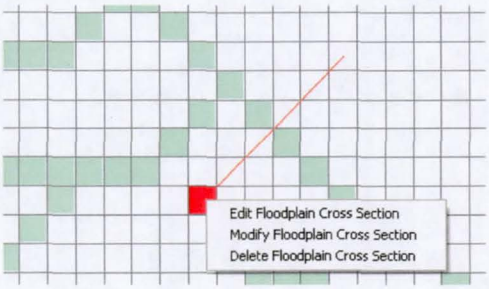
1. On the *Tool menu* click *Create Floodplain Cross Sections.*
2. The mouse pointer changes to a cross.
3. Locate and position the mouse over the first grid element in a line of grid elements that will constitute the cross section. Click on his first grid element of the cross section and then while holding the left mouse button down select the cross section grid elements as you move the mouse over the grid system. Cross section elements are joined by lines to visualize the cross section location. As the cross section development progresses from one element to the next, only the eight adjacent elements can be accessed. When the mouse pointer is near the center of the next element to be selected, a solid arrow appears to indicate that that element is the next one in the segment. The user can backup the pointer over the segment to unselect channel grid elements. No clicks are necessary, the users only needs to sweep the pointer over the desired elements. The cross section is completed when the user double-clicks with the left mouse button.



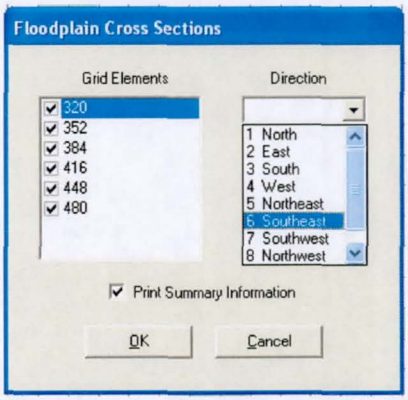
4. Cross sections must be consists of grid elements in a continuous straight line and can only be in the four directions as indicated below:



To edit the floodplain cross section grid elements, click on a cross section. A short-cut menu appears with three options as shown:



Select *Edit Floodplain Cross Section* to display the floodplain cross section dialog box:



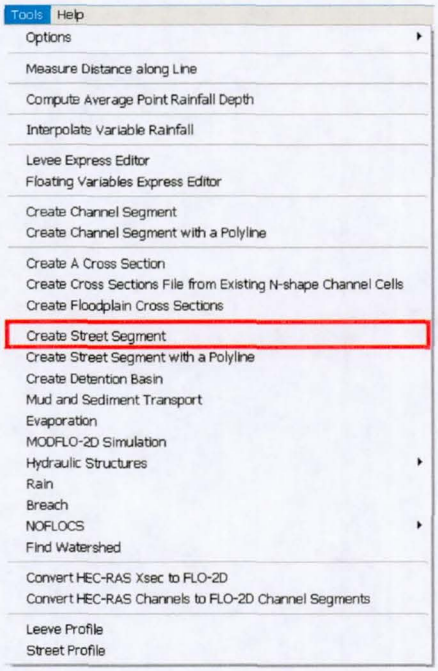
In the left data table are the grid elements that form the cross section. In the right data table of the dialog box, select the direction for discharge calculation. The cross section discharge will be computed for the selected flow direction that is perpendicular to the line of cross section grid elements. The model will create three output files:

CROSSQ.OUT - This file contains the element grid hydrographs for each of the floodplain elements in the cross section. The time and discharge are listed for each output interval.

CROSSMAX.OUT - This file lists the maximum discharge, maximum flow depth and time of occurrence for each grid element specified in the cross section analysis.

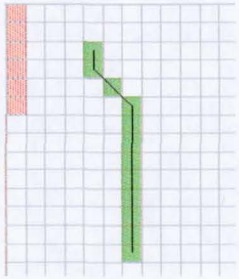
HYCROSS.OUT - The output interval time, top width, depth, velocity and discharge are listed for each cross section. The discharge passing the cross section of grid elements is compiled as a hydrograph. The cross section maximum discharge and the individual grid elements are written to the CROSSMAX.OUT file.

5.6.14 Create Street Segment Command (Tools Menu)



This command enables the creation of the streets in an existing *FLO-2D* grid system using point and click tools as described below:

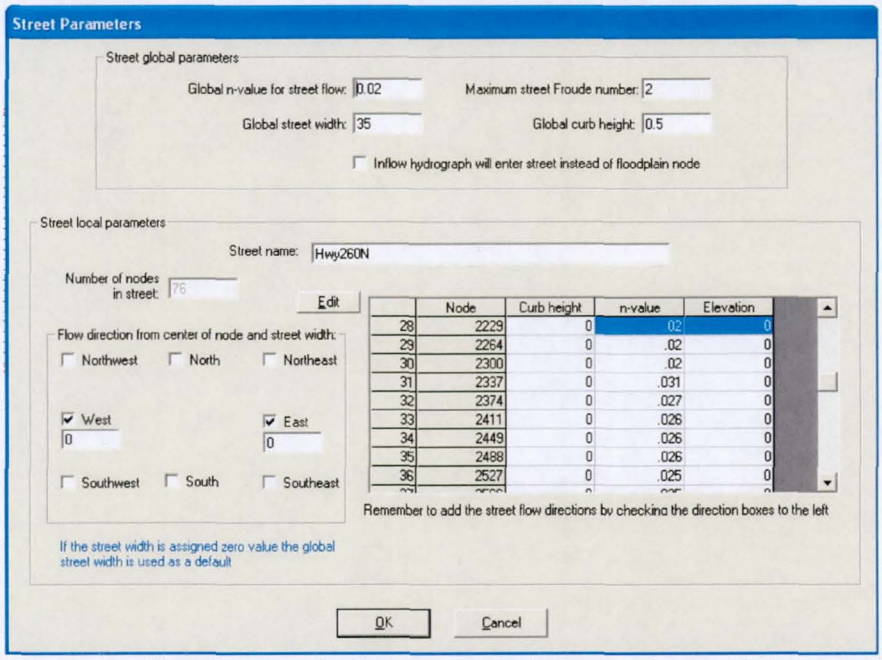
1. On the *Tool menu* click *Create Street Segment.*
2. The mouse pointer changes to a cross.
3. To add a new grid element to the street, click on first grid element of the street segment. Then while holding the left mouse button down, the system selects street elements as the user moves over the grid system. A background map or aerial photograph image can be used to locate the streets. The streets are identified by green elements and with a solid continuous line when been generated in the *GDS.*



As the street development progresses from one grid element to the next, only the eight adjacent elements can be accessed (see above figure). When the mouse pointer is located near the center of the next element to be selected, a solid line arrow will appear to indicate that the element will be added to the street. The user can backup the pointer over the segment to unselect street grid elements. No mouse clicks are necessary, it is only necessary to move the mouse pointer over the desired elements. To finish the street element selection, the user double-clicks with the left mouse button.

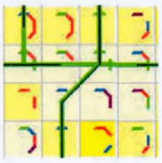
The street data assignment is not complete until the user assigns the street flow direction, street widths and n-values in the street editor dialog box. When the user clicks on an existing street, the street line will be colored in red and the following options appear:

To edit the street data, select *Edit Street Parameters* to display the Street Parameters dialog box:

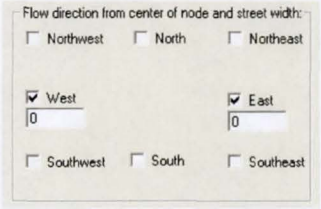


The Street Parameter dialog box can also be accessed by left mouse button click on a grid element with streets and clicking on the *Street Element* button. The street flow directions in the lower left of the dialog box are required data and must be assigned after completing the Create Street Option above (Task 3 above). Instructions for entering street follow:

1. *Global n-value for street flow* - Global n-value that is assigned to all the street elements.
2. *Maximum Street Froude number* - When the computed Froude number for the street flow exceeds this limiting street Froude number, the n-value is increased by 0.001 for that street element.
3. *Global street width* - Global assignment of street width (ft or m) for all street elements. This street width is superseded by individual street widths if the individual street width is greater than zero (see *Flow direction from center of node and street width* check boxes).
4. *Global curb height* - Global street curb height (ft or m). If the street curb height is exceeded by the flow depth, overland flow will result in the grid element containing the street. This value is used to assign a street curb height globally to all street elements.
5. *Inflow hydrograph will enter the streets rather than floodplain node* - Check this box to have the inflow hydrograph enter the street rather than the floodplain portion of the grid element.
6. *Curb height* - Optional curb height (ft or m) for an individual street element that supersedes the *global curb height*.
7. *Elevation* - Optional individual street element elevation (ft or m). This elevation will supersede the floodplain grid element elevation. If this value is zero, the model will assign the street elevation as grid element floodplain elevation minus the curb height.
8. *N-value* - Optional street n-value for the individual street segments within a given grid element. This value supersedes the *Global n-value for street flow*. If individual street n-value is zero, the global value will be assigned to the grid element street segment.
9. *Street name* - Character name of the street. Up to 15 characters can be used. The street name is not used in the *FLO-2D* model.
10. *Flow direction from center of node and street width* - Streets emanate from the center of the grid element in a star pattern of eight potential flow directions as show below.



When a given flow direction box is checked, a text box appears direct below the *flow direction check box* to input the optional grid element street width for that street flow direction ISTDIR:



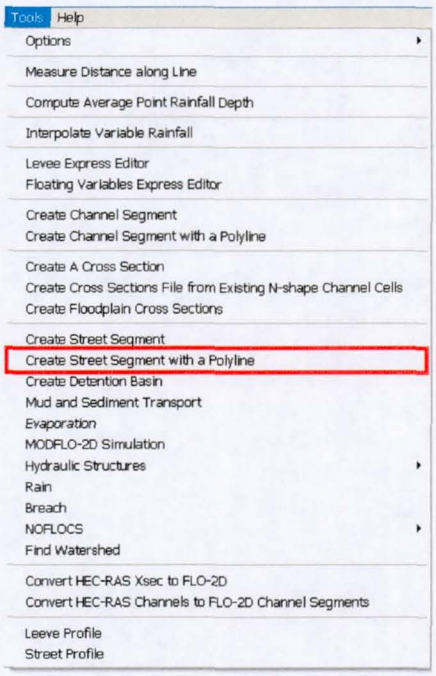
By setting the text box to zero, the *Global street width* will be assigned to that street segment.

Additional street data instruction comments can be found in the STREET.DAT file descriptions later in the manual. A few comments are highlighted here.

- Drawing the street outline with the Create Street Option does not complete the data assignment. You must click on the street and add the global street parameters, street flow directions and individual street element parameters if they are different from the global parameters.
- The street depth, width and n-values can be assigned globally for all the street grid elements. The street depth, width and elevation can also be assigned for individual grid elements in the street local data form by setting these values greater than zero.
- If the street elevation is different from the representative grid elevation assigned in the FPLAIN.DAT file, it should be specified in the Street Local Form otherwise the street elevation will be the floodplain elevation minus the curb height. These elevations are then used to determine the street slope.
- The street width should be less than the side of the grid element. The overall floodplain surface area of the grid minus the streets surface area must be at least 5% of the original surface area (grid element width squared). If the street surface area consumes the entire grid element surface area, consider leaving out the smaller, less significant streets, reducing the street widths or transferring one street to a neighboring grid element.

The street is assumed to extend from the center of the grid element to the grid element boundary in the eight flow directions. A street that crosses the entire grid element is assigned two street sections and directions.

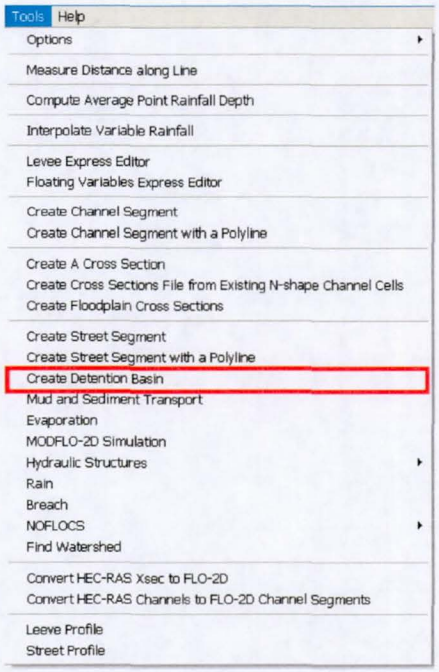
5.6.15 Create Street Segment with a Polyline Command (Tools Menu)



With this command you can create a street segment using simple mouse point and clicks. First click on *Create Street Segment with a Polyline* command; the cursor will change to a cross and as you click the mouse, the channel segment will be progressively created. To complete the segment, just double-click.

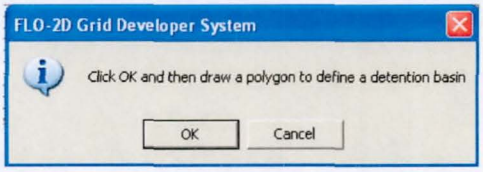
5.6.16 Create Detention Basin Command (Tools Menu)

This command will help to design detention basins for flood mitigation. The detention basin is created by adjusting the grid element topography to reflect the detention basin depression. No special *FLO-2D* components or data are necessary to utilize the detention basin feature.

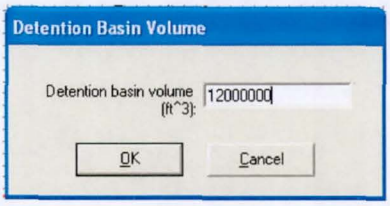


The procedure to create a detention basin depression in the grid system topography is as follows:

1. The user selects the *Create Detention Basin* Command.
2. The system displays the following message:

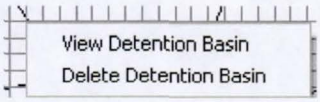


3. The user draws the polygon that defines the detention basin perimeter. The polygon is closed by double clicking on the last vertex of the polygon.
4. The *GDS* will then request for the detention basin volume (ft^3 or m^3):

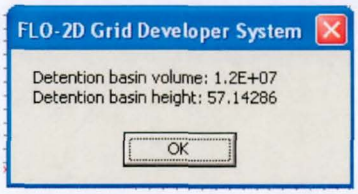


5. Input the desired volume and click *'OK'*. The system will subtract a uniform depth (*Detention basin height*) from all the elements included in the detention basin polygon until the detention basin volume is matched. The floodplain element elevations are lowered to account for the *Detention basin volume*.

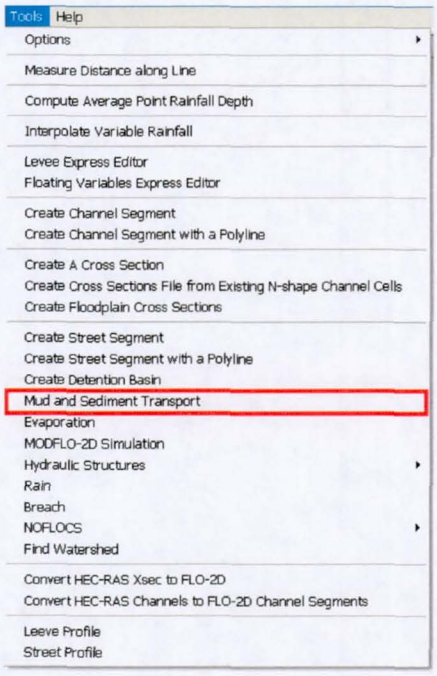
If the user clicks on the detention basin polygons, the following menu appears:



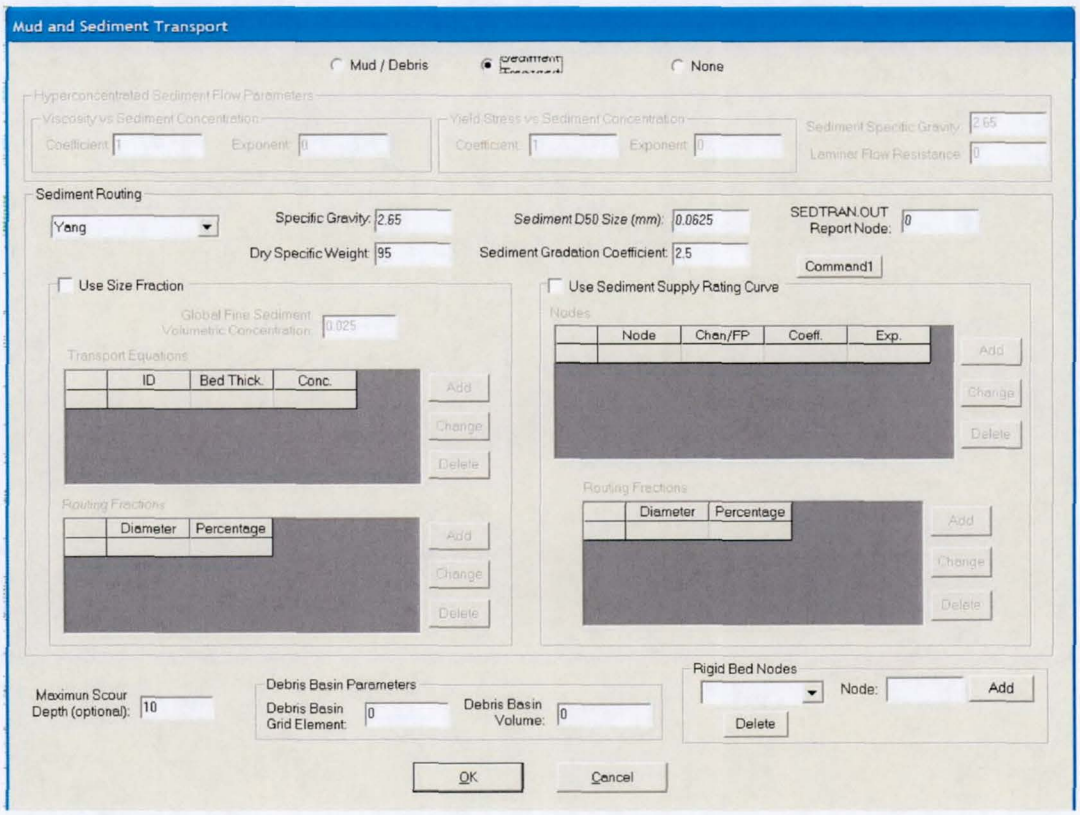
When you click on *View Detention Basin* the following message is displayed:



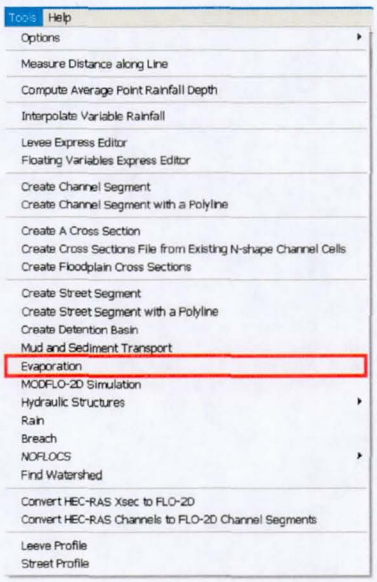
5.6.17 Mud and Sediment Transport Command (Tools Menu)

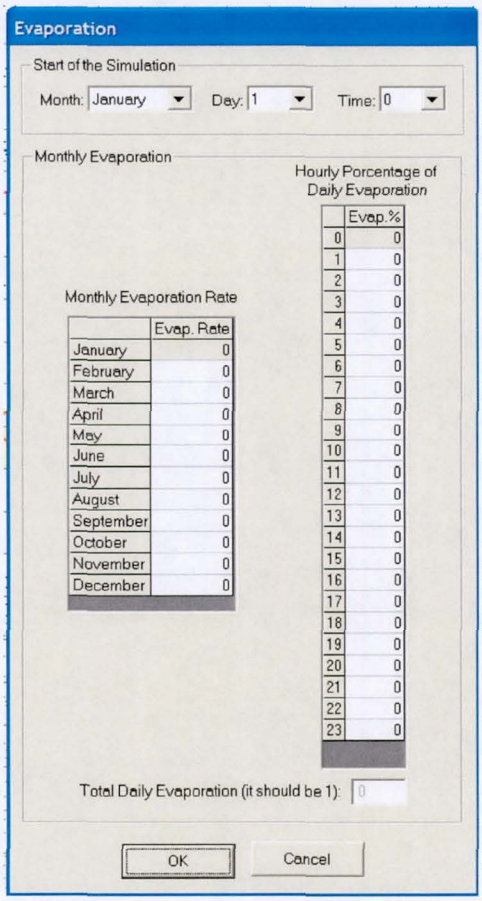


This command allows entering mud flow or sediment transport data using the following dialog box.

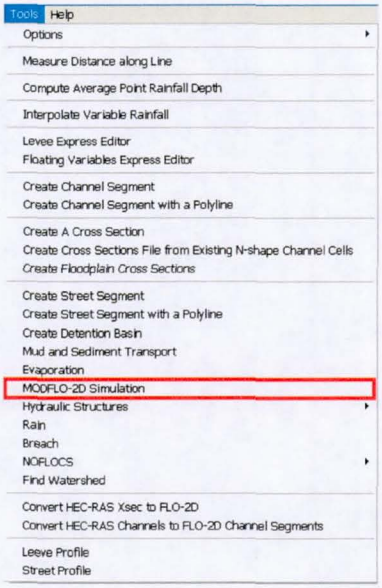


5.6.18 Evaporation Command (Tools Menu)

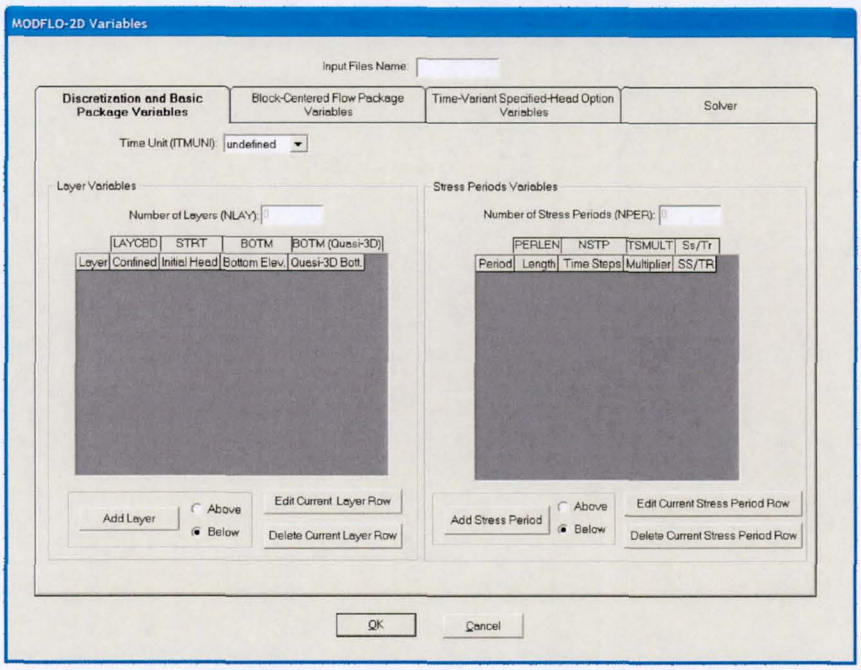




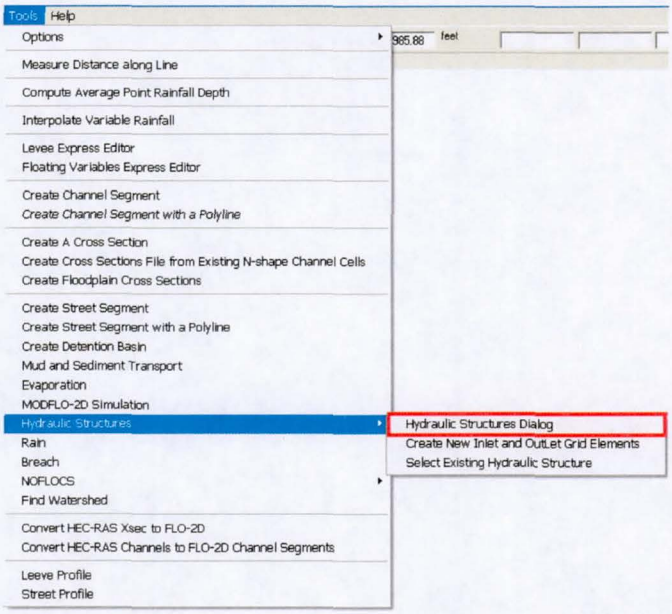
5.6.19 *MODFLO-2D* Simulation Command (Tools Menu)



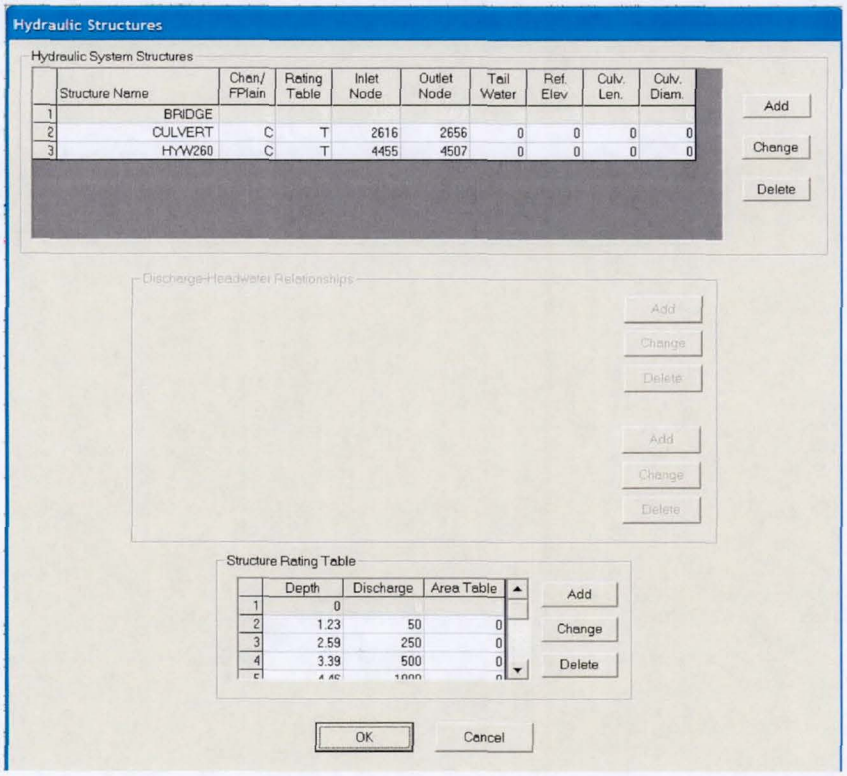
This command allows entering data to simulate surface and groundwater flow using the *MODFLOW* integration. There following tabbed dialog box provides input fields for a subset of *MODFLOW* data. A separate manual explain in full the capabilities of the *FLO-2D MODFLOW* link, also called *MODFLO*-2D.



5.6.20 Hydraulic Structures Command (Tools Menu)



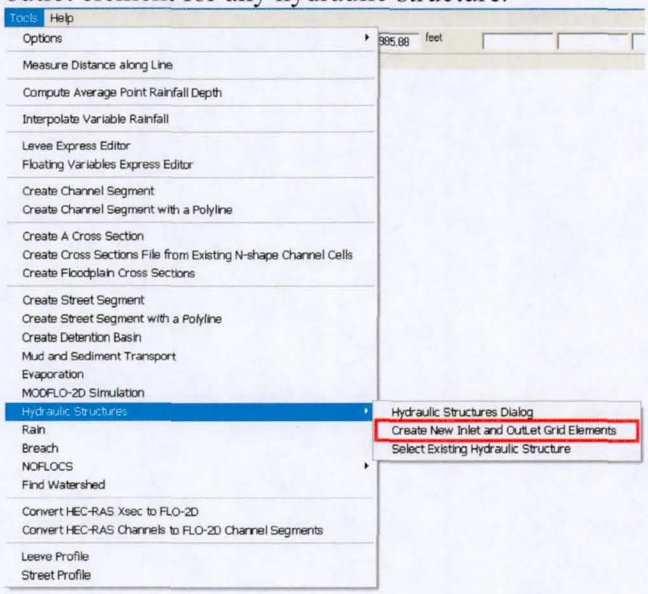
The Hydraulic Structures command allows entering data required to simulate culverts, bridges and any other hydraulic structure that can be represented with a rating table or rating curve.



	Structure Name	Chan/ FPlain	Rating Table	Inlet Node	Outlet Node	Tail Water	Ref. Elev	Culv. Len.	Culv. Diam.
1	BRIDGE								
2	CULVERT	C	T	2616	2656	0	0	0	0
3	HYW260	C	T	4455	4507	0	0	0	0

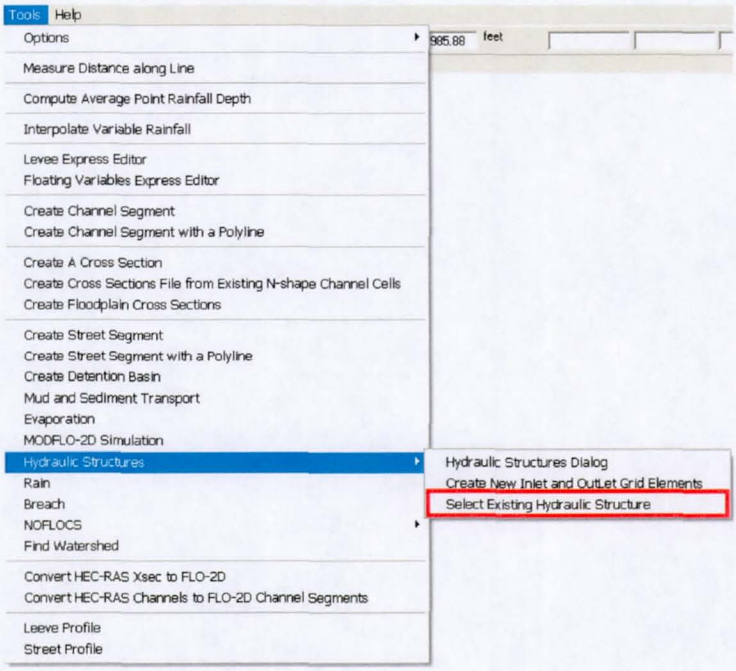
	Depth	Discharge	Area Table
1	0		
2	1.23	50	0
3	2.59	250	0
4	3.39	500	0

The create *Inlet and Outlet Grid elements* Command is used to interactively enter the required inlet and outlet element for any hydraulic structure.

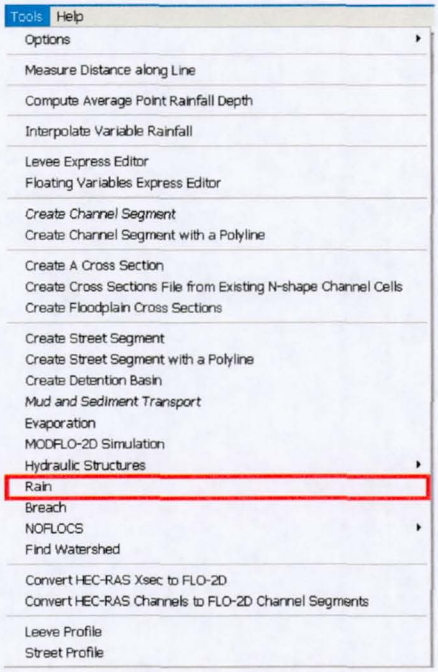


The *Select Existing Hydraulic Structure* command allows editing data for a previously

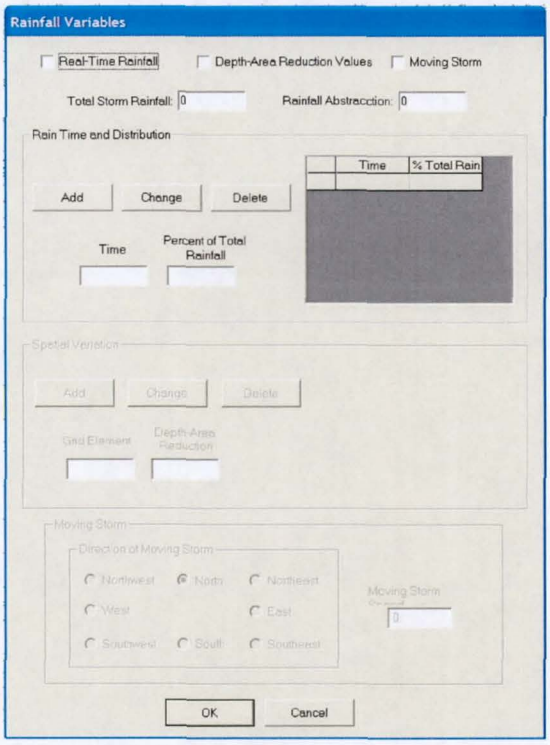
defined structure.



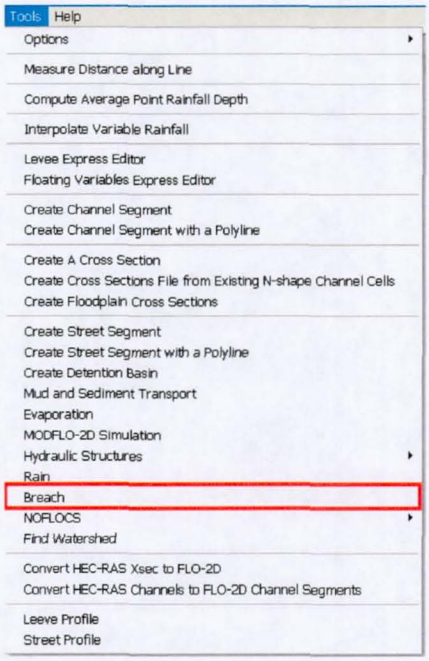
5.6.21 Rain Command (Tools Menu)



The Rain command allows entering data to simulate rainfall input over the FLO-2D grid.



5.6.22 Breach Command (Tools Menu)



This command is used to enter levee breach data for the computed breach option.

Global Breach Data

Breach Variables

Levee Fragility Curve | **Global Breach Data** | Individual Breach Data

Global Geometry Data

Upstream Face Slope:	2	Maximum Bottom Width:	0
Downstream Face Slope:	2	Maximum Top Width:	0
Core Average Face Slope:	0	Pipe Bottom Elevation:	0
Crest Width:	5	Weir Coefficient:	3.05
Crest Length:	0		

Global Core Data

Mean Sediment Size:	0	Angle of Internal Friction:	0
Porosity:	0	Cohesive Strength:	0
Unit Weight:	0	Core Sediment Gradient:	0
Manning's n-value:	0		

Global Shell Data

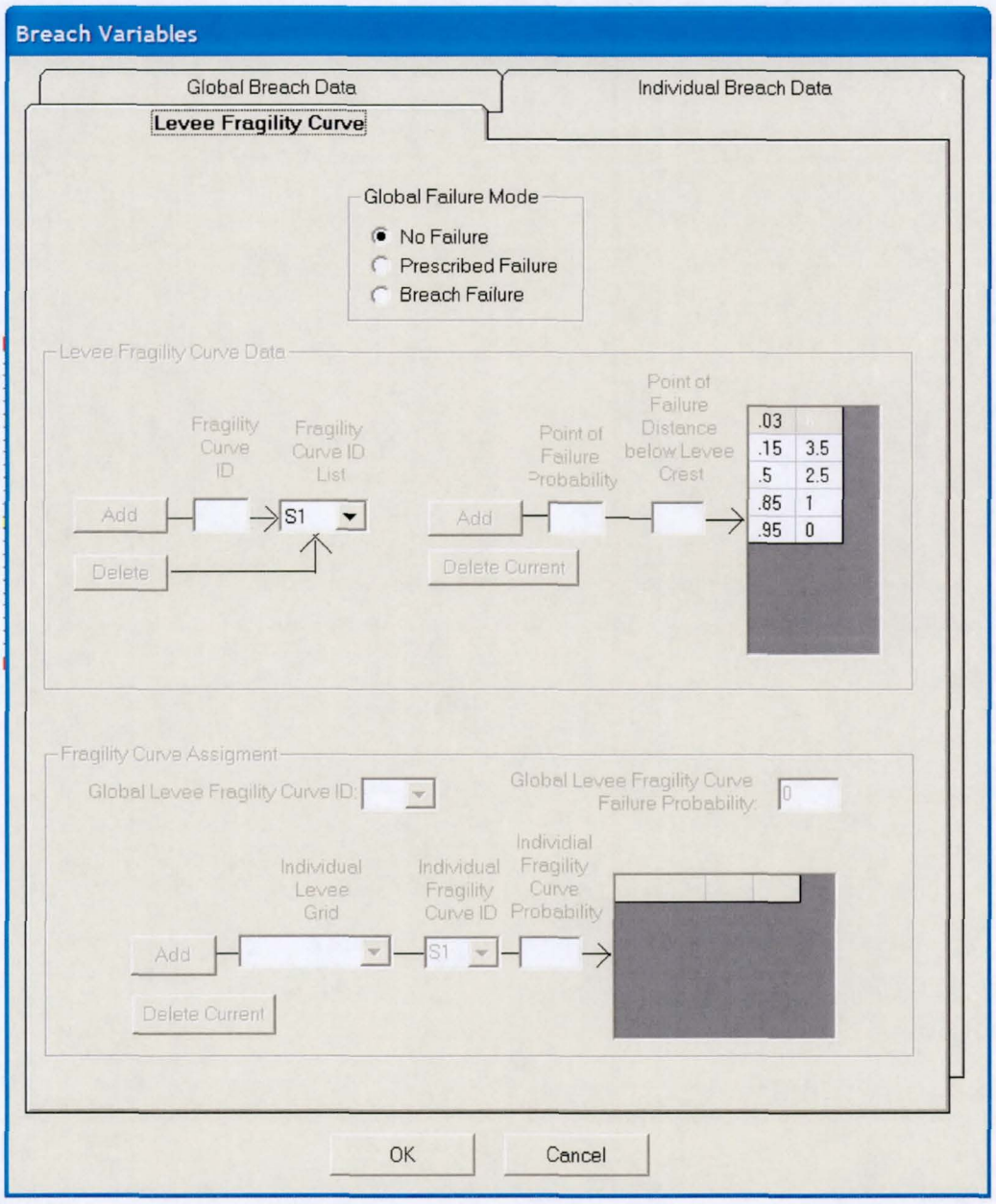
Mean Sediment Size:	0.25	Angle of Internal Friction:	30
Porosity:	0.4	Cohesive Strength:	100
Unit Weight:	100	Shell Sediment Gradient:	0
Manning's n-value:	0.06		

Global Miscellaneous Data

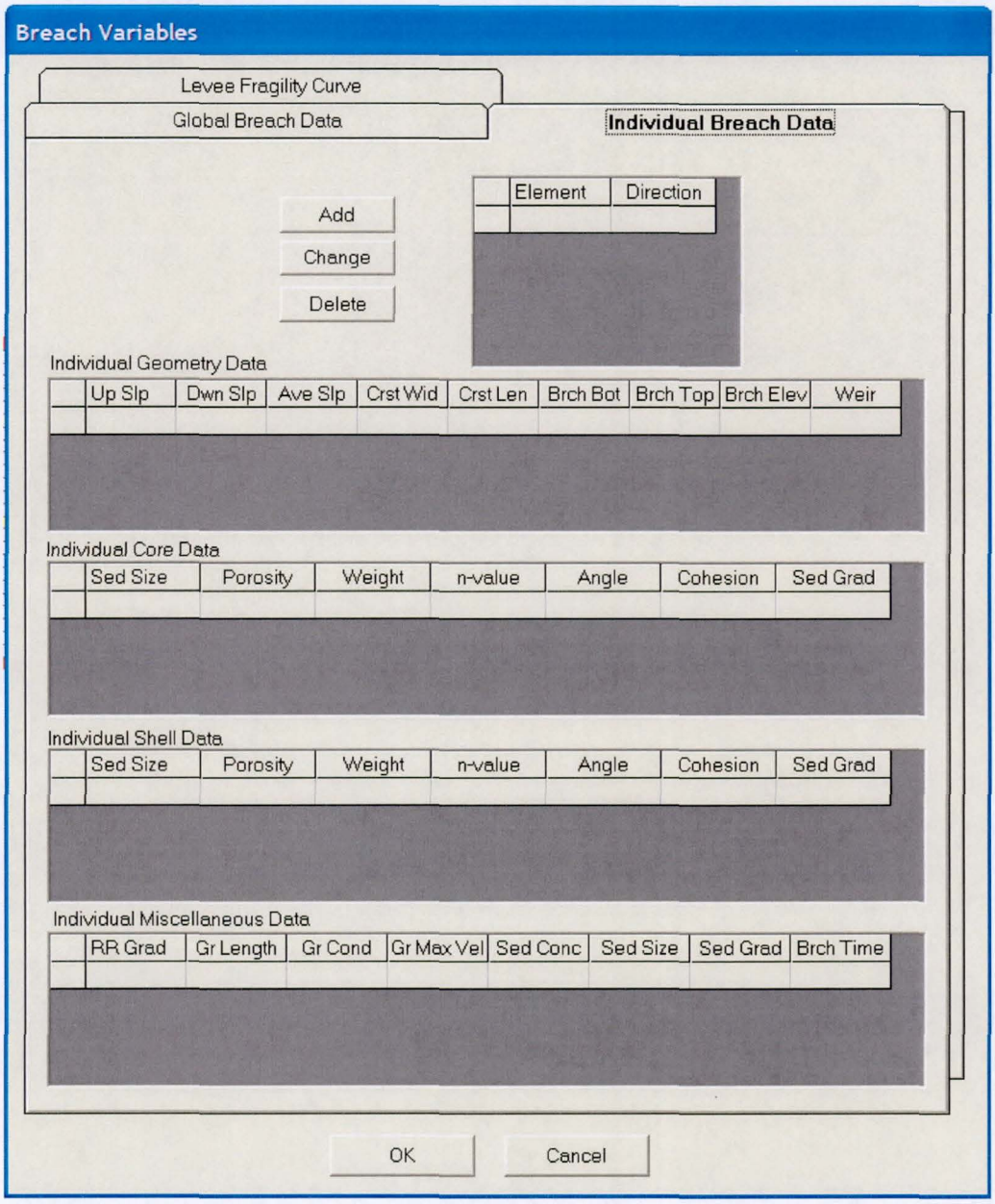
Initial Breach Width/Depth Ratio:	1	Max Sediment Conc:	0
Average Grass Length:	0	Riprap Size:	0
Downstream Grass Condition:	0	Riprap Gradient:	0
Maximum Grass Velocity:	0	Time to Initiate Failure:	0.1

OK | Cancel

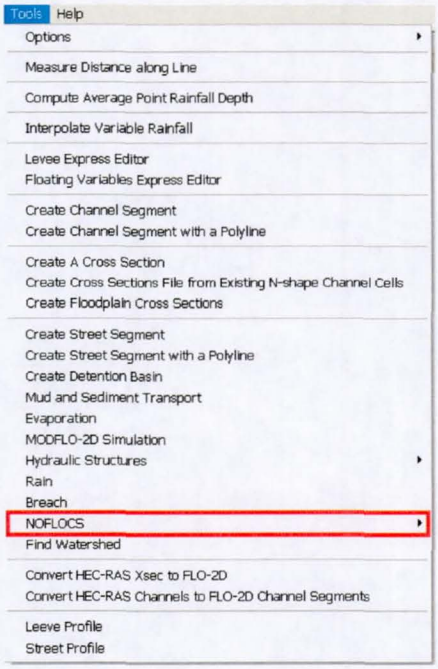
Levee Fragility Curve Data

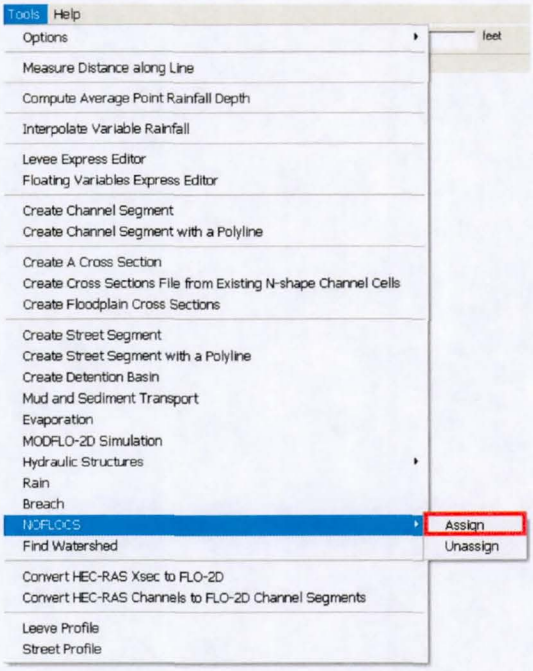


Individual Breach Data

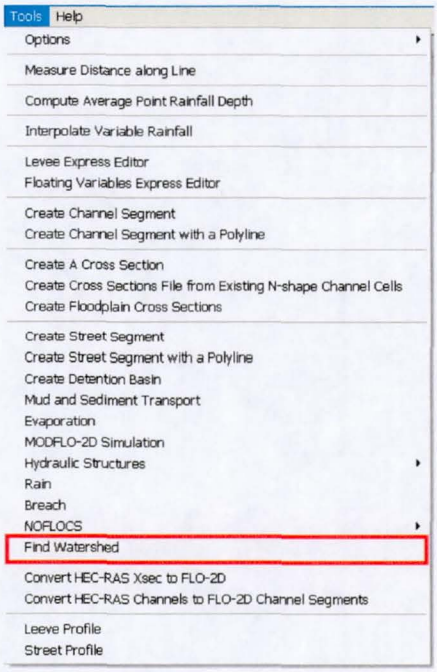


5.6.23 NOFLOCS Command (Tools Menu)

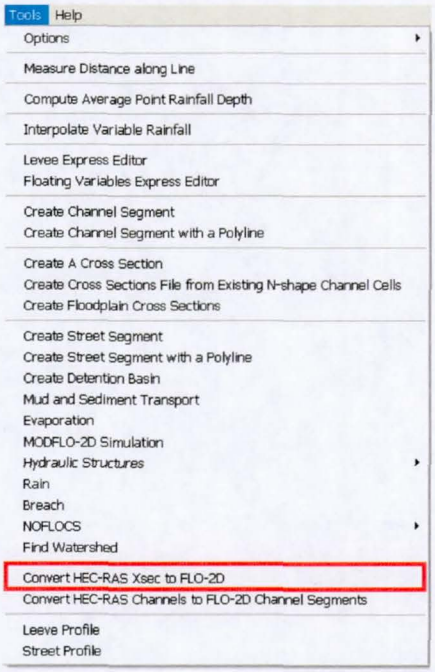




5.6.24 Find Watershed Command (Tools Menu)



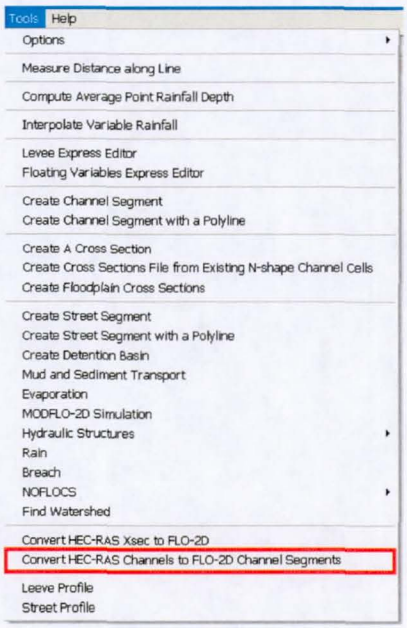
5.6.25 Convert *HEC-RAS* Xsec to *FLO-2D* Command (Tools Menu)



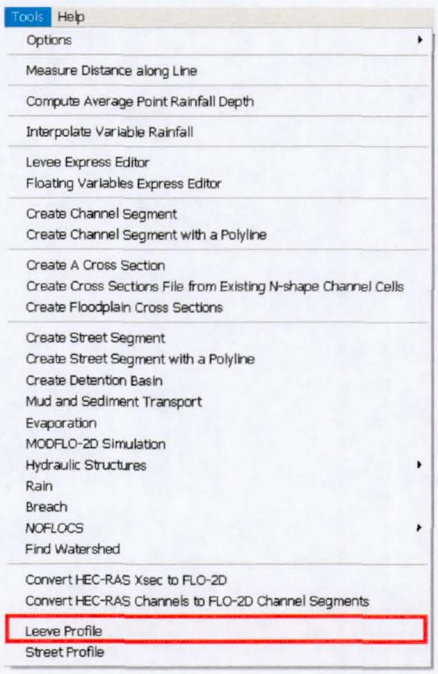
This feature will read cross section data available in *HEC-RAS* format, convert it to *FLO-2D* format and save it to XSEC.DAT files. The *GDS* reads the *HEC-RAS* file, locates the cross

section key words, extracts the x, y, z data from each cross section and reformat the cross section data for the XSEC.DAT file. When this process is completed, another file dialog box appears to let the user select the location of the output XSEC.DAT file.

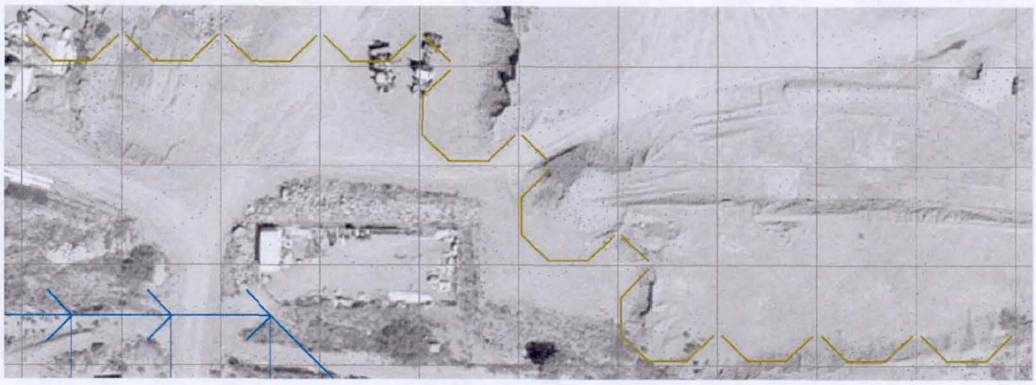
5.6.26 Convert *HEC-RAS* Channels to *FLO-2D* Channel Command (Tools Menu)



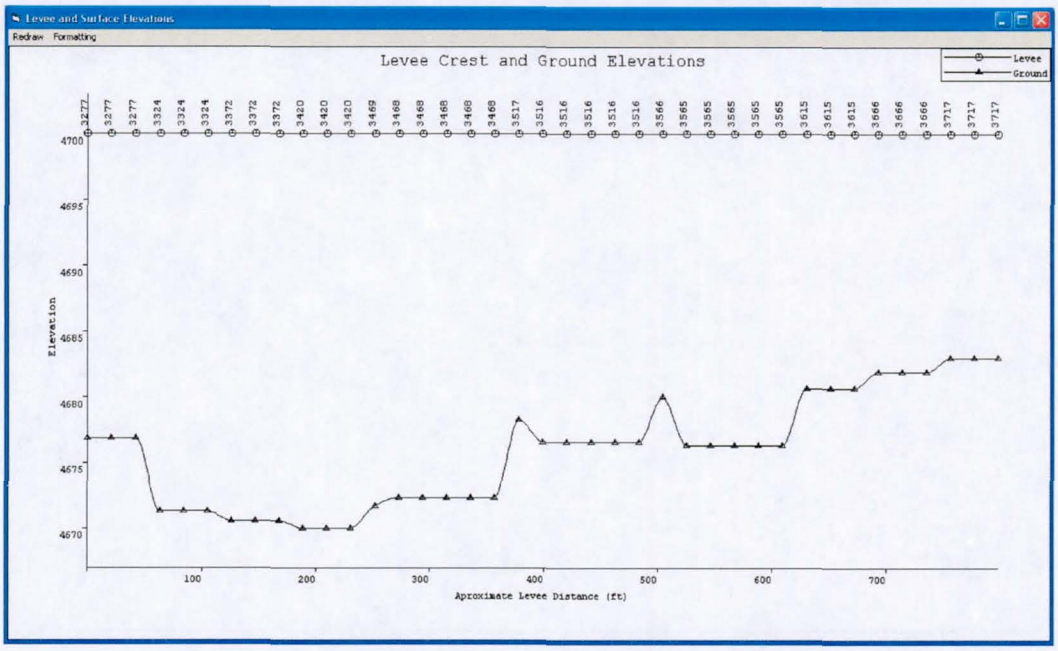
5.6.27 Levee Profile Command (Tools Menu)



This tool will display levee crest elevation profile of a selected levee segment along with the corresponding grid element ground elevations. To activate this tool, click on the command *Levee Profile* and then click on one of the grid elements that contain a levee. *GDS* will determine the continuity of the levee and change the levee color to dark green to identify the selected elements as shown in the figure below.

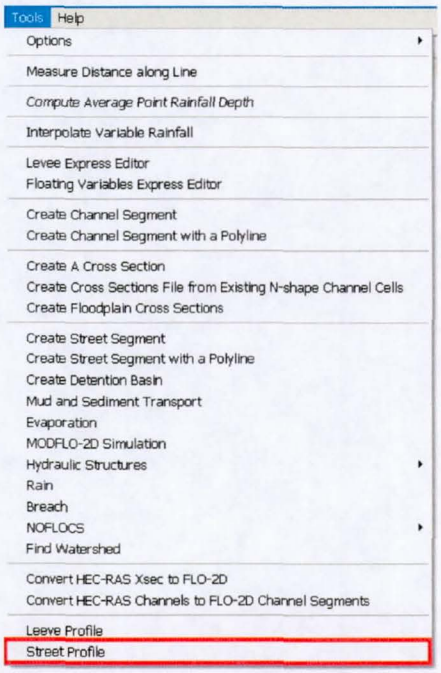


Then *GDS* will display the levee profile as shown in the following plot:

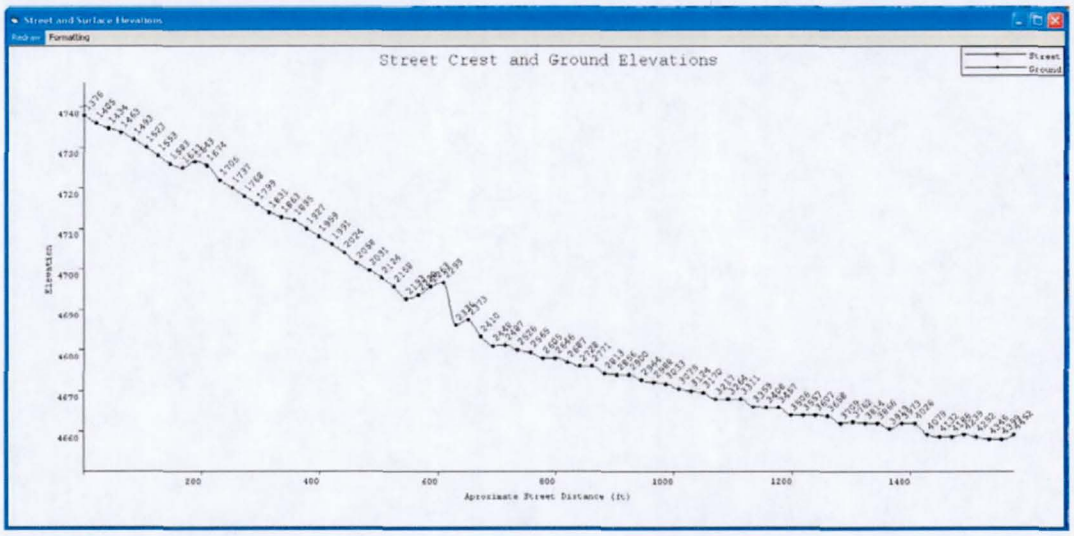


Using this *Formatting* Menu on the tool bar, the *GDS* will display the following dialog box so that you can customize the plot including the minimum and maximum elevations (Y-axis), legend position and angle of labels.

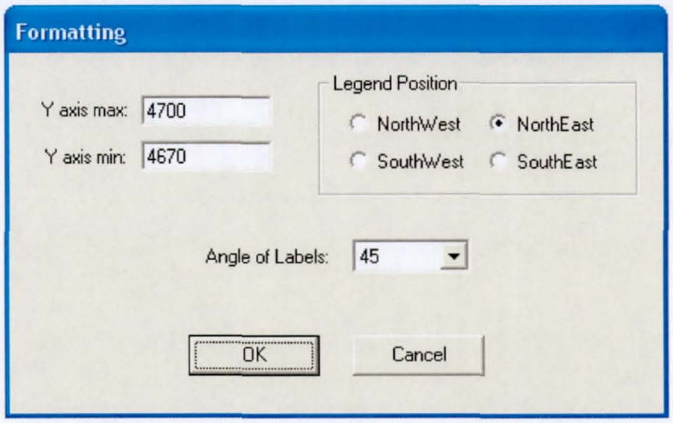
5.6.28 Street Profile Command (Tools Menu)



This command will create a street and ground elevation profile. To access this tool, click on the *Street Profile* Command and click on any street element. The *GDS* will display the street profile as shown:



Similar to the *Levee Profile Command*, use the *Formatting* command on the tool bar to display the following dialog box to customize the plot including the minimum and maximum elevations (Y-axis), Legend position and angle of labels.

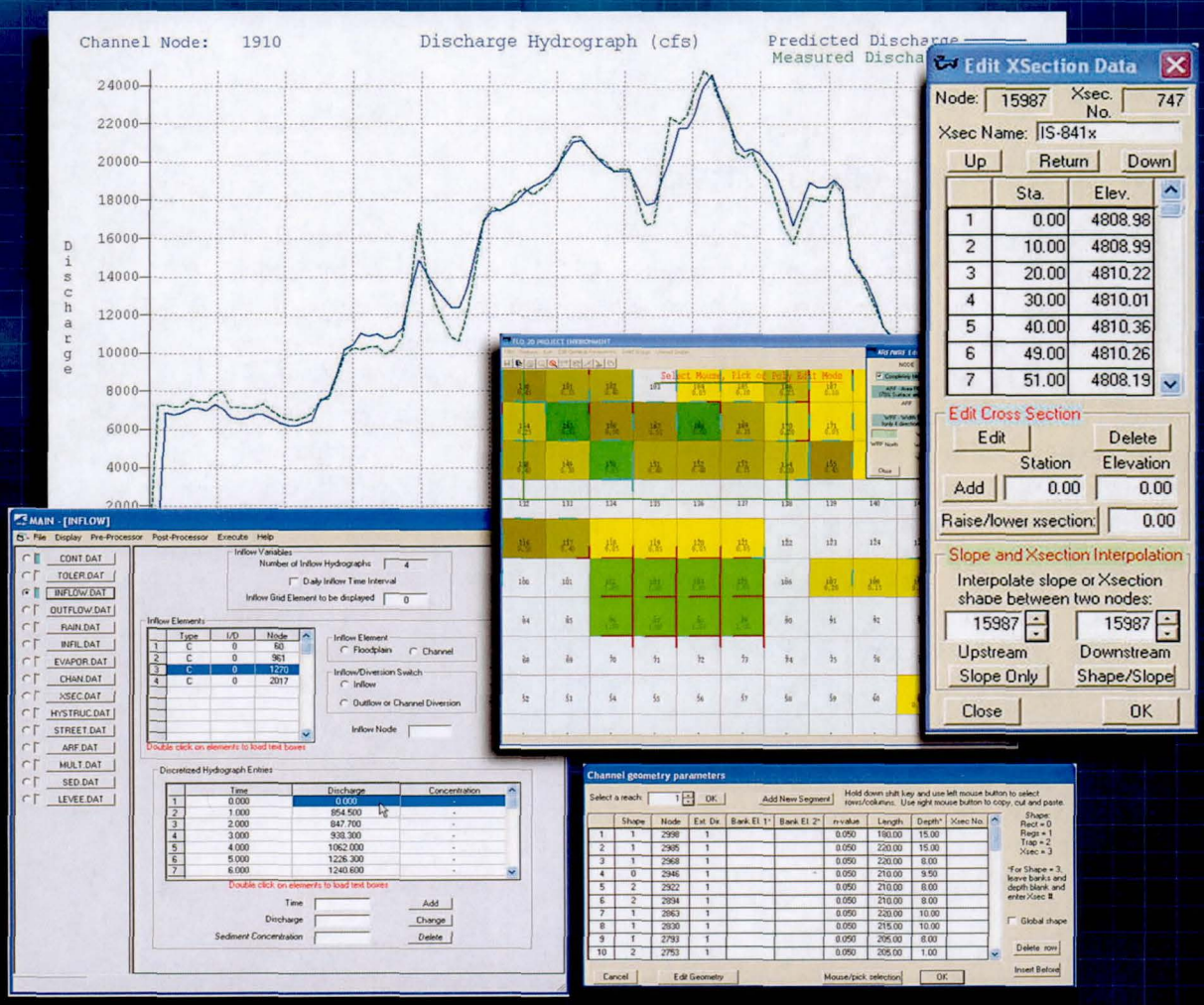


5.7 Help Command:

The Help Command accesses the online help system that contains essentially all the information in this *GDS* User Manual.

FLO-2D

Data Input Manual



PREFACE

FLO-2D INPUT DATA OVERVIEW

This manual describes the FLO-2D Pro Model data input parameters and their format. The FLO-2D data consists of a series of ASCII files organized by model components. A flood model starts with routing a hydrograph over an unconfined floodplain surface. This model can then be expanded with channel flow or other components. Flood simulation detail can be enhanced by adding rainfall, infiltration, hydraulic structures, levees, mudflows, sediment transport, rills and gullies, storm drains, buildings and flow obstructions. These components are initiated through on-off switches found in the control data file (CONT.DAT). If the component options are "turned on", then the appropriate data files must be created.

To conduct a basic FLO-2D flood simulation, six data files must be created. These files can be automatically generated (error free) using the grid developer system (GDS) program. The six required files for basic overland flow simulation are:

- FPLAIN.DAT
- CADPTS.DAT
- CONT.DAT
- TOLER.DAT
- INFLOW.DAT
- OUTFLOW.DAT.

The FPLAIN.DAT and CADPTS.DAT files contain the topographic data. CONT.DAT and TOLER.DAT contain control and numerical stability data and consist of only a few lines of data. INFLOW.DAT contains the inflow hydrograph and OUTFLOW.DAT contains outflow nodes and optional outflow water

surface control. There are two new data files in the Pro model to assist with GIS and CADD program integration: TOPO.DAT (coordinate data and elevation) and MANNINGS_N.DAT (cell and roughness n-value). The user is encouraged to start simple with a basic overland flow simulation and build the flood detail into the model one component at a time to observe the effects of each feature. Pre-processor programs such as the Grid Developer System (GDS) and PROFILES that facilitate developing and graphically editing the data files.

There are several ways to edit the FLO-2D data files. Since the data files are written in ASCII format, they can be edited in any ASCII editor such as Microsoft WordPad®, TextPad®, UltraEdit®, and others. The GDS program enables multiple selections of grid elements to edit spatially variable data with mouse point and click commands. The PROFILES program can be used to edit channel and cross section data.

There are also two ways to run a FLO-2D simulation once the data files are constructed. The Pro model can be initiated from the GDS and a FLO-2D flood simulation can also be started by copying the FLOPRO.EXE file into a project subdirectory and double-clicking on the file name FLOPRO.EXE in a Windows browser. When the model is running the user has the option of graphically viewing the flood progression over the grid system. An inflow hydrograph and the rainfall temporally distribution is also displayed. Upon completion of the flood simulation, there are post-processor programs (MAPPER, MAXPLOT, PROFILES and HYDROG) that will assist in reviewing the results.

This Data Input Manual includes descriptions of the processor programs, data variables and file format, and output files. Each data file description contains a list of variables, variable definitions and instructional comments. The instruction comments at the end of each file description provide hints for data organization, range of data values and data limitations. For a discussion of the physical processes being simulated please refer to the FLO-2D Reference Manual.

Contents

FLO-2D INPUT DATA OVERVIEW i

Chapter 1 FLO-2D Installation and Getting Started 1

1.1 General 1
1.2 FLO-2D Installation 1
1.3 Un-installing the FLO-2D Software 2
1.4 Getting Started 2
1.5 Creating a Channel 7
1.6 Troubleshooting Guidelines 11
1.7 Model Component Considerations 16

Chapter 2 Creating and Editing with the GUI 17

2.1 FLO-2D Tutorials 17
2.2 FLO-2D Example Flood Simulations 19
2.3 FLO-2D PowerPoint Presentations 21
2.4 Other Help Documents 22
2.5 Metric Option 22

Chapter 3 Pre-Processor Programs 23

3.1 Introduction 23
3.2 GDS 23
3.3 PROFILES 23

Chapter 4 Input Data File Description 28

4.1 General 28
4.2 List of Program File Units 29
Data Files 30
4.3 Data Files 31
CONT.DAT 32
TOLER.DAT 40
TOLER.DAT 42
FPLAIN.DAT 45
MANNINGS-N.DAT 48
TOPO.DAT 51
INFLOW.DAT 54

TOLSPATIAL.DAT

OUTFLOW.DAT 57
RAIN.DAT 62
INFIL.DAT 65
EVAPOR.DAT 74
CHAN.DAT 76
CHANBANK.DAT 87
XSEC.DAT 89
HYSTRUC.DAT 91
STREET.DAT 99
STREET.DAT 101
ARF.DAT 103
MULT.DAT 106
SED.DAT 109
LEVEE.DAT 117
FPXSEC.DAT 123
BREACH.DAT 126
FPFROUDE.DAT 138
SWMMFLO.DAT 140

Chapter 5 Output Files and Options 144
5.1 Basic Guidelines for Using Output Files 144
5.2 Output File Descriptions and Options 145

Chapter 6 Post-Processor Programs 159
6.1 HYDROG 159
6.2 MAPPER ++ 162
6.3 MAXPLOT 163
6.4 PROFILES 165

Chapter 7 Debugging and Trouble Shooting the Data Files 169
7.1 Basic Trouble Shooting Guidelines 169
7.2 List of Common Data Errors 170
7.3 Runtime Errors 171
7.4 Troubleshooting Guid 172

LIST OF TABLES

Table 1.1. Grid System Size 4
Table 1.2. Partial List of Data Files by Unit Number 12
Table 2.1. English/Metric 21
Table 4.1. List of FLO-2D Files and Unit Numbers 30
Table 7.1. List of Most Common Data Error 170

CHAPTER 1

FLO-2D INSTALLATION AND GETTING STARTED

1.1 GENERAL

This manual can be used to help create the FLO-2D input data and review the output data. It has chapters on getting started, resources, pre processor programs, data files, output files, post processor programs and troubleshooting. Chapter 4 can be a useful tool when learning to build data files. It breaks down each data file into a set of variables and gives a definition and instructional comments for the data files.

FLO-2D is recommended for Windows 7 or 8 computers with 64-Bit operating systems and steady state hard drives. To generate and edit the data files, the Grid Developer System (GDS) processor program is used. The GDS facilitates assigning spatially variable data and automated interpolated data from shape files. PROFILES is used to edit channel geometry data. Data files can also be edited using an ASCII text editor such as UltraEdit© or NotePad©.

1.2 FLO-2D INSTALLATION

FLO-2D is compatible with recent versions of the MS-Windows™ operating systems including Windows 7 and 8©. The FLO-2D Pro model has been compiled for 64 bit multi-core processor computers. You cannot run the Pro Model Executable on a 32-bit computer. The GDS can be used on 32-Bit or 64-Bit operating systems. Recommended minimum computer requirements are at least 4 GB RAM and 16 GB of available hard disk space. To load the Pro model system onto the computer hard drive, unzip the installation file and double click the file FLO-2D-PRO-Setup.exe file. Follow the installation instructions as they appear in the dialog boxes on the screen. It is necessary to select a subdirectory for the

installation. The default is C:\PROGRAM FILES\FLO-2D PRO. The FLO-2D model, and all the processor programs are loaded into the FLO-2D folder. The FLO-2D resource files are saved to the FLO-2D Documentation folder under C:\Users\Public\Public Documents. These files include helpful resources such as user manuals, example projects, lessons and PowerPoint presentations and instructional handouts.

1.3 UN-INSTALLING THE FLO-2D SOFTWARE

Remove the FLO-2D program and all of its attendant software from the computer with the Windows system uninstall command. When removing the model, if the option to keep shared DLL/OCX files is displayed, do not remove them from the computer. The GDS uses several shared DLLs that may be essential for other applications. To completely remove the FLO-2D files, delete the FLO-2D Folder from whatever location it was installed. If FLO-2D folder contains project files, move them or back them up before un-installing or re-installing the model.

1.4 GETTING STARTED

Updates

When starting a new FLO-2D project, first visit the website www.flo-2d.com/download and download any executable updates. New features are frequently added to the model. Do not hesitate to notify us of any apparent programming bugs or problems that are encountered and we will address as soon as possible. Program revisions are listed on the web site in the FLO-2D Pro Model Revisions document.

Tutorials and Lessons

The lesson book Workshop Lessons is located in the FLO-2D help folder (FLO-2D Documentation\flo_help). It has step-by-step instruction of how create and edit detailed flood components.

> Hint:
> When sending data files for tech support, zip together only *.DAT files and send them to contact@flo-2.d.com

Seeking Assistance – Technical Support

Send technical support questions by e-mail to contact@flo-2d.com. If there is a specific problem that needs to be resolved, zip the data files (only the *.DAT files, no output files *.OUT) and attach them to the e-mail along with a brief description of the problem and the project. Before sending the files, try to reduce the problem to its simplest form by turning off all those components that are not part of the problem. For example, if the problem involves channel volume conservation, turn off the streets, buildings and levees and run the simulation again to determine if the problem still persists. Try to identify when the problem is first observed during the simulation (review SUMMARY.OUT) so that it is not necessary to run the entire

simulation to replicate the error. Often this process alone will help to resolve the problem.

Questions regarding a project application are considered to be technical consulting and outside the scope of data input technical support. If assistance is needed on a project, reasonable consulting fees can be discussed to provide guidance and oversight.

Hydrology, Base Mapping and DTM Points

DTM Data

To start a FLO-2D model, define the project area and compile available mapping, imagery and digital terrain model (DTM) data which might consist of LiDAR data, shape files, contour maps or digital elevation model DEM data. The imagery and DTM points must have the same coordinate system. The most common formats for digital imagery are *.tif, *.sid and *.jpg files and the images must have corresponding world files (e.g. *.tfw, *.sdw and *.jgw). If photogrametric or LiDAR data are not available, DEM data can be used. Elevation data formats that are accepted by the GDS are ASCII x y z data sets and ArcGIS elevation shape files.

Hint:
Use any ASCII editor such as NotePad™ to edit the data files.

Hydrologic data

Hydrologic data for a flood simulation can include both rainfall and discharge hydrographs. The rainfall runoff in a FLO-2D watershed model can be modeled as the project result or it can be used to generate inflow flood hydrographs for downstream flood routing. In either case the hydrologic data should be carefully reviewed because the area of inundation is determined by the flood volume.

Hint:
An alternate method to run the model is to copy the FLO.EXE file into the project folder and double click on it from a browser.

Floodplain and channel detail

If river channels, bridges, culverts, buildings and streets are to be simulated, the user must be able to locate these features with respect to individual grid elements. Aerial imagery and shape files are used for this purpose. Additional data may be required for these components including bridge and culvert rating curves or tables, streets width and curb height, and river cross section surveys.

Estimate the project area

To create a computationally efficient model, it is best to attempt to minimize the grid system around the project area. The project computational domain (or grid system) can be outlined using the aerial photography. The grid system boundary should be located so that the project area it is not affected by either inflow or outflow conditions. The inflow and outflow nodes should be considered as non-essential nodes (sources and sinks) and these should be located away from the project area.

If the project area is relatively small compared to the entire hydrologic basin that may need to be modeled, more than one FLO-2D simulation could be considered. A coarse grid system can be established for watershed or river system and a more detailed grid system created for the local project area where flood detail may be important. The outflow from the course grid system will constitute the inflow to the detailed grid system.

Selecting the grid element size

Once the overall project area has been identified, estimate the grid system size (as a rough rectangle) and determine the approximate number of grid elements that would be required for different size square grid elements such as 50 ft, 100 ft, 200 ft, etc. The grid element size will control how fast the FLO-2D flood simulation will run.

TABLE 1.1. GRID SYSTEM SIZE

Number of Grid Elements	Model Simulation Speed
< 50,000	Fast (minutes)
50,000 – 250,000	Moderate (<12 hours)
250,000 – 1,000,000	Slow (> 12 hours)
> 1,000,000	Very Slow (> 1 day)

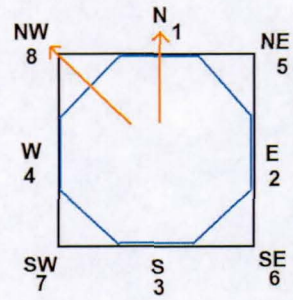
To help with the grid element size selection, the following criteria are suggested based on a rough estimate of peak discharge. The peak discharge Q_{peak} divided by the surface area of the grid element A_{surf} should be in the range:

$$Q_{peak}/A_{surf} < 10.0 \text{ cfs/ft}^2$$

or in metric:

$$Q_{peak}/A_{surf} < 3.0 \text{ cms/m}^2$$

The closer Q_{peak}/A_{surf} is to 1.0 cfs/ft^2 (0.3 cms/m^2), the faster the model will run. If the Q_{peak}/A_{surf} is much greater than 1.0 cfs/ft^2 or 0.3 cms/m^2, the model will run more slowly. After the grid element size has been selected, proceed with establishing the grid system using the GDS. There are GDS workshop lessons to assist in getting started on a new project.



Start simple, then add detail

The first flood simulation for any project will be a simple overland flow model upon which a more detailed flood simulation will be gradually built.

A suggested order of component construction is as follows:

- Rainfall/Infiltration
- Channels
- Levees
- Streets
- Buildings
- Hydraulic Structures (culverts, weirs and bridges)
- Storm Drains (SWMM model)
- Multiple Channel (rills and gullies)
- Mud and debris flows/sediment transport

As new components are added to a model and tested, other components switches can be turned off in the CONT.DAT file.

FLO-2D routes flows in eight directions as shown in the sidebar figure. The four compass directions are numbered 1 to 4 and the four diagonal directions are numbered 5 to 8. Some components such as levees are placed on boundaries of the grid element. The grid element boundaries constitute an octagon for components associated with the boundary.

Hint:

Basic Data Files:

- **FPLAIN.DAT**
- **CADPTS.DAT**
- **CONT.DAT**
- **TOLER.DAT**

Optional

- **INFLOW.DAT**
- **OUTFLOW.DAT**

Saving data

When creating or editing the data files, it is suggested that to save the data files frequently and use one folder for testing a project and another one for editing a project. It is suggested that you save the data files after working on each component.

Building a Project

Create a Project Folder

Start by creating a subdirectory for the project data files and import the DTM data base files, map images and aerial photos.

Build the Project Files

Use the GDS to build a grid system. The data files can be graphically created in the GDS. Follow the GDS "Getting Started" lesson.

Run the FLO-2D model

Hint:
Test initial model runs with a simple steady state inflow hydrograph.

The required data files for a basic overland flood model are:

- FPLAIN.DAT
- CADPTS.DAT
- CONT.DAT
- TOLER.DAT
- INFLOW.DAT
- OUTFLOW.DAT

The INFLOW.DAT and OUTFLOW.DAT files are optional but are typically necessary for most applications. Run a FLO-2D simulation by:

i) GDS - click on '*Run FLO-2D*' command in the File menu.

ii) Copy the '*FLOPRO.EXE*' file in the project folder and double click it.

Some General Guidelines

Data Input

When the data format seems confusing, review the data files provided in the Example Projects subdirectory of the FLO-2D folder.

File Management

The output files in the project folder will be overwritten during subsequent model runs. To save any output files that might be overwritten, rename the file or create a new project folder, copy all the *.DAT files into it and then run the new flood simulation in that folder.

Graphics Mode

To graphically view the floodwave progression during the simulation, run the data files in graphics mode. This switch is set in the GDS by clicking File/Run FLO-2D to activate the control dialog box. Then check the Graphics Display mode and the Run button.

Things to check when creating the data files:

Grid System

The grid system should begin with grid element #1 and have no missing grid element numbers. There should be no dangling grid elements connected only by a diagonal. This may occur when manually

Inflow/Outflow Nodes

Inflow and outflow nodes should not have other components assigned to them such as hydraulic structures, streets, ARF's, etc. Outflow nodes should not be doubled up. Use a single line of outflow nodes.

1.5 CREATING A CHANNEL

Overview

Review the CHAN.DAT file description in the Data Input Manual. It includes a description of all channel variables and instructions on how to apply them. In the FLO-2D model, river flooding using the channel component is simulated as one-dimensional, depth averaged flow. Each channel element is represented by either rectangular, trapezoidal or station/elevation cross section. Simulating river flow requires the following information and data:

- Location of the channel with respect to the grid system;
- Channel roughness;
- Length of channel within the grid element;
- Channel cross section or geometrical data.

Channel slope is computed as the mean bed elevation difference between the channel elements. Channel elements must be contiguous to be able to share channel discharge. The key to accurately simulating channel flow is to assess the relationship between the slope, flow area and roughness.

Hint:

The channel bank on each side of the rive can have a unique elevation. If the two bank elevations are different, the model automatically assigns the channel into two elements even if the channel would fit into one grid element.

CREATING THE CHANNEL DATA FILE

The procedure for creating a river channel simulation is as follows (see the various Channel and PROFILE Lessons for step-by-step instructions):

Select Channel Cross Sections

River cross section survey data is organized in the XSEC.DAT file with

a station and bed elevation format. A cross section may represents one or more channel elements and each channel element is assigned a cross section in the CHAN.DAT. For channel design projects, a rectangular or trapezoidal cross section may be appropriate. Cross section data can be automatically extracted from HEC-RAS models or cross sections can be cut from a DTM data base.

> Warning:
> **Channel elements must be contiguous to share discharge. Each channel element should appear only once in the CHAN. DAT file.**

Locate the Channel Element with Respect to the Grid System

Use the GDS to identify the left bank channel element (see the GDS Channel Lesson). The channel left elements must be neighbors to enable flow through the system. The left bank channel elements can be automatically assigned by importing a geo-reference HEC-RAS channel data file.

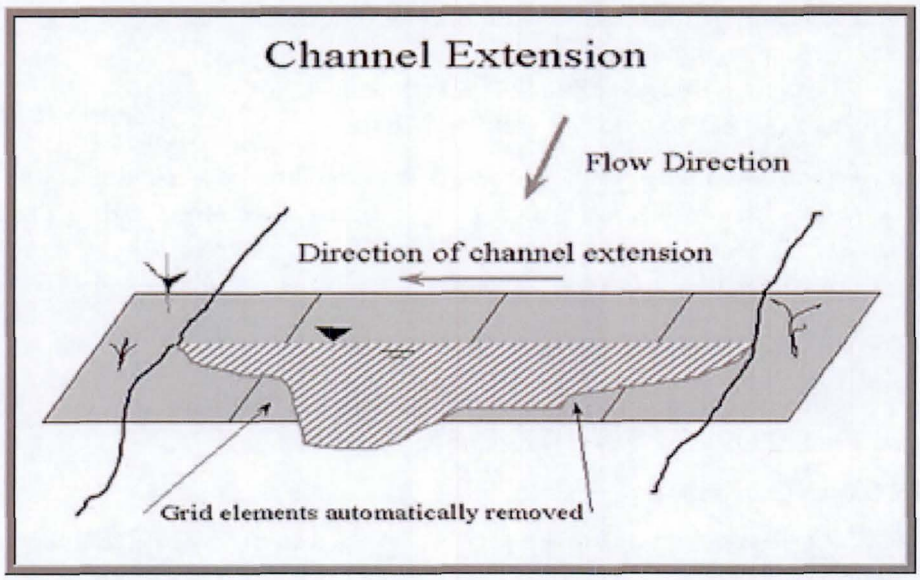
Assign the Preliminary Channel Data

The GDS will list the selected channel elements in a dialog box. Channel data such as shape, channel element number, roughness n-value, channel length and cross section number can be assigned in the editor dialog box.

Define the Right Bank Element

The channel width can be larger than the grid element. For example, a channel may be 1000 ft wide while the grid element is only 200 ft wide. The left and right bank elements can be separated by several grid elements. The channel routing component interacts with the right and left bank elements to share discharge with the floodplain. Each bank element can have a unique top-of-bank elevation. Channel bank elements are listed in CHANBANK.DAT.

FIGURE 1.1. CHANNEL EXTENSION



Assign the Cross Section Number

The surveyed cross section number in XSEC.DAT must be assigned to the corresponding channel element in CHAN.DAT. Assign a zero (0) value to the channel cross section numbers that associated with a surveyed cross section. Typically there are a limited number of cross sections and many channel elements. This procedure has been also automated. Before interpolating the cross sections to all the channel elements, the channel profile will look like a staircase. After clicking on the GDS 'Interpolation' button each channel element will have a unique cross section and bed elevation and the cross sections numbers in XSEC.DAT and CHAN.DAT will correspond and will be renumbered from top to bottom starting with cross section number 1. Both the slope and cross section shape will be interpolated between those channel elements with assigned cross sections.

Assign the Channel length Within the Grid Element

The channel length (XLEN) within a grid element should be estimated. The GDS calculates the XLEN automatically. The channel lengths are then summed and reported by GDS for each segment. The river center-line distance can be estimated with the GDS *'Measure Distance along a Line'* tool. The individual XLEN values can then be adjusted so that the segment total length matches the measured center line distance.

Adjust the Channel Bed Slope and Interpolate the Cross Sections

The cross section geometry and slope can be re-interpolated between any two channel elements in the PROFILES program. The result of this interpolation is an adjusted cross section shape and bed slope. The assigned surveyed cross sections can retain their original shape and elevations. The bed slope for rectangular and trapezoidal can also be adjusted.

Assign the Manning's n-value.

Initially a uniform default Manning's n-value is assigned to all the channel elements. Using the limiting Froude number (FROUDC) in Line 1 of the CHAN.DAT file, spatially variable n-values can be adjusted. The n-value should represent a composite flow resistance for the entire channel including bed irregularities, obstructions, vegetation, variation in channel geometry, channel expansion and contraction, potential rapidly varying flow and variable river planform. Poor selection of n-values (particularly underestimating n-values) or failure to provide spatial variation in roughness can result in numerical surging.

Additional Channel Data Instructions

The user has several other options for setting up the channel data file including grouping the channel elements into segments, specifying initial flow depths, assigning confluence pairs, identifying channel elements that don't share discharge with the floodplain (NOEXCHANGE), assigning limiting Froude numbers and specifying depth variable n-value adjustments. These options are discussed in the CHAN.DAT file description. Instructional comments follow:

1. Dividing the channel into segments may simplify working with a river system and reviewing the results. For example, a tributary can be a segment. Organize the channel elements and segments from upstream to downstream.

2. If there are channel elements that will not share flow with the floodplain, set NOEXCHANGE equal to the channel element. An example is a underground culvert which will not receive any floodplain inflow.

3. To improve the timing of the floodwave progression through the system, a depth variable roughness can be assigned on a segment basis. The equation for the channel element roughness n_d as function of flow depth is:

$$n_d = n_b \, rc \, e^{-(r2 * depth/dmax)}$$

where:

n_b = bankfull discharge roughness
$rc = -1/e^{-r2}$
depth = flow depth
dmax = bankfull flow depth
r2 = roughness adjustment coefficient prescribed by the user (0. to 1.2).

This equation assumes that the assigned roughness is based on bankfull discharge. If the user assigns a roughness adjustment coefficient r2 (ROUGHADJ in CHAN.DAT) for a given reach, the roughness will increase with a decrease in flow depth. The higher the coefficient r2, the greater the increase in roughness. A typical value of ROUGHADJ is 0.4.

FYI:

The channel length within a grid element (XLEN) is defined by the position of the channel within the grid element. If the channel is positioned in a straight line across the cell, XLEN is equal to the width of the grid element.

XLEN = X * Grid Element Width

X: 1.0 1.207 1.414

Channel Data Dependencies

There are several channel data dependencies with variables in other files.

CONT.DAT:

ICHANNEL - Channel 'on' or 'off' switch
NOPRTC - print options.

INFLOW.DAT:

IDEPLT - channel inflow node hydrograph to be plotted on the screen at run-time. Assign IFC = 'C' for channel inflow hydrographs.

OUTFLOW.DAT:

KOUT - channel outflow nodes

Channel Output

Channel output can be reviewed in several ways. The channel output data is written to a series of ASCII output files including: BASE.OUT, HYCHAN.OUT, CHAN-MAX.OUT, DEPCH.OUT and others. The HYDROG program will display a plot of the hydrograph of each channel element. It also has a routine to compute average hydraulic conditions (flow area, bed shear stress, hydraulic radius, velocity, etc.) in a channel reach covering several channel elements that the user selects. The PROFILES program can be applied to review the water surface profile, spatial variation in peak discharge, mobile bed profiles, or the cross section geometry changes associated with scour and deposition. Finally MAXPLOT and MAPPER++ will graphically plot the channel maximum flow depths and velocities.

1.6 TROUBLESHOOTING GUIDELINES

Data Errors

Data input errors may result in the automatic termination of a simulation run along with an error message. The error message will report a 'Unit' number that is associated with the data input file that contains an error. Referring to these numbers may help you to debug input data errors.

TABLE 1.2. PARTIAL LIST OF DATA FILES BY UNIT NUMBER

Unit No.	File Name	Unit No.	File Name
9	TOLER.DAT	39	SED.DAT
10	CADPTS.DAT	50	OUTFLOW.DAT
30	CONT.DAT	52	STREET.DAT
31	FPLAIN.DAT	57	LEVEE.DAT
32	RAIN.DAT	68	HYDROSTRUC.DAT
33	INFIL.DAT	85	XSEC.DAT
34	INFLOW.DAT	89	RAINCELL.DAT
36	CHAN.DAT	119	CHANBANK.DAT
37	ARF.DAT	120	FPXSEC.DAT
38	MULT.DAT	180	WSTIME.DAT

A more complete list of file unit numbers can be found in the Chapter 4 of the Data Input Manual.

Troubleshooting: Is the flood simulation running OK?

There are several indicators to help you identify modeling problems. The most important one is volume conservation. The FLO-2D results should be reviewed for volume conservation, surging, timestep decrements, and roughness adjustments with limiting Froude numbers.

Volume Conservation

Any hydraulics model that does not report on volume conservation should be suspected of generating or losing volume. A review of the SUMMARY. OUT file will identify any volume conservation problems. This file will display the time when the volume conservation error began to appear during the simulation. Typically a volume conservation error greater 0.001 percent is an indication that the model could be improved. The file CHVOLUME. OUT will indicate if the volume conservation error occurred in the channel routing instead of the overland flow component. Components should be switched 'off' one at a time and the model simulation run again until the volume conservation problem disappears. This will identify which component is causing the difficulty. Some volume conservation problems may be eliminated by slowing the model down (decreasing the timesteps) using the numerical stability criteria. Most volume conservation problems are an indication of data errors.

Surging

It is possible for volume to be conserved during a flood simulation and still have numerical surging. Numerical surging is the result of a mismatch between flow area, slope and roughness. It can cause an over-steepening of the floodwave identified by spikes in the output hydrographs. Channel surging can be identified by discharge spikes in the CHANMAX.OUT file or in the HYDROG program plotted hydrographs. Predicted high maximum velocities indicate surging. To identify floodplain surging, review the maximum velocities in the MAXPLOT or Mapper post-processor program. You can also review the VELTIMEC.OUT (channel) or VELTIMEFP. OUT (floodplain) files for unreasonable maximum velocities. Surging can be reduced or eliminated by adjusting (lowering) the stability criteria (DEPTOLFP or WAVEMAX in TOLER.DAT) thus decreasing the timesteps. If decreasing the timesteps fails to eliminate the surging, then individual grid element topography, slope or roughness should be adjusted. This can be accomplished in the GDS for floodplain flow. For channel flow, the PROFILES program can be used to make adjustments. Increasing the flow roughness will generally reduce or eliminate flow surging. For channel surging, abrupt transitions in flow areas between contiguous channel elements should be avoided. Setting a lower limiting Froude number for a channel reach may also help to identify the problem.

Sticky Grid Elements

When the flood simulation is running slowly, the TIME.OUT file can be reviewed to determine which grid elements are causing the most timestep decreases ('sticky elements'). TIME.OUT lists the top twenty floodplain, channel or street elements that caused the model to slow down. The file also lists whether the timestep decreases occur with the percent change in depth, Courant criteria or dynamic wave stability criteria. Adjustments can be made in the stability criteria to more equably distribute the timestep decreases. The model is designed to advance and decrement timesteps, so there have to be grid elements listed in the TIME.OUT file. If one or two grid elements have significantly more timestep decreases than the other elements listed in the file, the attributes of the 'sticky' grid elements such as topography, slope or roughness should be adjusted. The goal is to make the model run as fast as possible while avoiding numerical surging.

If a floodplain element is causing most of the timestep decreases, check the SURFAREA.OUT file to determine how much surface area is left in the floodplain element for flood storage. If the floodplain element contains a channel bank, there may be very little surface area left for flood storage. This will cause the model run slowly with exchanges the flow between the channel and floodplain. To fix this problem:

- Remove other components from the channel bank element including streets or ARF values.
- Shorten the channel length (XLEN in CHAN.DAT). This will increase the surface area in the channel bank floodplain elements.
- Decrease the channel cross section width in the PROFILES program.

Limiting Froude Numbers

There is a unique relationship between floodwave celerity and average flow velocity described by the Froude number that should not be violated during numerical flood routing. This is a physical relationship between the kinematic and gravitation forces. To use the limiting Froude number, estimate a reasonable maximum Froude number for your flood simulation and assign the value to either FROUDL (floodplain), FROUDC (channels) or STRFNO (streets) variables. When the computed Froude number exceeds the limiting Froude number, the n-value is increased by a small value (~ 0.001) for the next timestep. This change in grid element n-value helps to create a better match between the slope, flow area and n-value during the simulation. When the limiting Froude number is no longer exceeded, the n-value is gradually decreased to the original value. The changes in the n-values during the simulation are reported in the ROUGH.OUT file. For the next FLO-2D simulation, grid element n-value adjustments can be made using the n-values reported in ROUGH.OUT. The maximum n-values are also reported in FPLAIN.RGH, CHAN.RGH and STREET.RGH files that are created at the end of a simulation. These (*.RGH) files can then be renamed to data input files (*.DAT) for the next flood simulation (e.g. FPLAIN.RGH = FPLAIN.DAT).

Reviewing the results

FLO-2D results include the maximum area of inundation as displayed by the maximum flow depth, temporal and spatial hydraulic results, channel or floodplain cross section hydrographs and peak discharges. The Mapper++ program can used to review maximum flow depths, water surface elevations or velocities. The results can be plotted as either line contours or shaded contours in Mapper++. Look for any maximum velocities or flow depths that are unreasonable. This may be an indication of numerical surging.

The FLO-2D flood simulation can be terminated at any time during the run by clicking Exit on the toolbar. The simulation will terminate after the current timestep is completed and the output files are generated and saved. This enables the user to check if the flood simulation is running poorly (e.g. too slow or not conserving volume) and the simulation can be stopped without losing the opportunity to review the output data.

Hint:

Use the limiting Froude numbers (i.e. FROUDL), ROUGH.OUT file and *.RGH files to spatially vary the grid system n-values. This will calibrate the model to a reasonable maximum Froude number and help eliminate numerical surging.

Make some adjustments

The following data file adjustments may improve the simulation and speed up the model:

Spatial Variation of n-values

The most common cause of numerical surging is underestimated n-values. Typical n-values represent steady, uniform flow. Spatial variation of n-values will affect the floodwave progression (travel time) and reduce surging, but may not significantly impact the area of inundation (especially for longer flood durations). Focus on the project area when adjusting n-values and review TIME.OUT and ROUGH.OUT to complete the n-value revisions.

Edit Topography

The interpolation of DTM points to assign elevations to grid elements is not perfect even when the GDS filters are applied. It may be necessary to adjust some floodplain grid element elevations when you review the results. MAXPLOT and Mapper++ can be used to locate grid elements with unreasonable flow depths that may constitute inappropriate depressions. Floodplain depressions can sometimes occur along a river channel if too many floodplain DTM points located within the channel.

Floodplain Surface Area Reduction

The distribution of flood storage on the grid system can be influenced by assigning area reduction factors (ARF's) to represent loss of storage (i.e. buildings). For large flood events, the assignment of individual grid element ARF values will usually have minor impact on the area of inundation. For local flooding detail, individual grid element ARF assignments may be justified.

Channel Cross Section Adjustments

Typically a surveyed cross section will represent five to ten channel elements. Selecting a cross section to represent transitions between wide and narrow cross sections requires engineering judgment. Use the PROFILES program to interpolate the transition between surveyed cross sections.

Channel Slope Adjustments

Adverse channel slopes can be simulated by FLO-2D. Smoothing out an irregular slope condition over several channel elements to represent reach average slope conditions may speed up the simulation. Cross sections with scour holes can result in local adverse slopes that misrepresent the average reach conditions. Review the channel slope in PROFILES.

Street Flow

High street velocities may cause numerical surging and slow the simulation down. Assign reasonable limiting street Froude numbers to adjust the street n-values.

Model Calibration and Replication of Flood Events

Estimating flood hydrology (both rainfall and flood hydrographs) can be difficult when replicating historical floods. To match measured flood stages, high water marks or channel discharges, first determine a reasonable estimate of the flood volume, then concentrate on the model details such as n-values, ARF's and street flow. Flood volume is more important to flood routing than the peak discharge.

1.7 MODEL COMPONENT CONSIDERATIONS

Street Flow

Streets may convey or store only a small portion of the total flood volume, but may be important for distributing the flow to remote areas of the grid system. Street flow is simulated as a shallow rectangular channel with curbs. Street width and n-values are spatially variable. Streets are important to flood distribution in urban areas.

Levees

Levees and levee failure can be an important detail for floodplain projects. Levees are assigned to grid element boundaries with a crest elevations. Levee failure can include piping, overtopping and collapse. There is a levee and dam erosion component in FLO-2D.

Rainfall and Infiltration on Alluvial Fans

Alluvial fan surfaces can be as large as the upstream watershed. Fan rainfall can contribute a volume of water on the same order of magnitude as the inflow flood hydrograph at the fan apex. Infiltration losses can also significantly effect flood-wave attenuation. Infiltration losses can be calibrated by adjusting the hydraulic conductivity. Spatial variable hydraulic conductivity can be assigned in the GDS.

Sediment Bulking of Flood Hydrographs

An alluvial fan will have geomorphic features that identify the watershed potential for generating mudflows. For mudflow simulation, sediment concentration can be assigned in the INFLOW.DAT file. For desert alluvial fans with a sand bed, sediment concentrations in flood events can reach 15% by volume. For concentrations less than 20% by volume, the flow will behave like a water flood. The primary effect of increasing the sediment concentration, in this case, is to bulk the flow volume.

CHAPTER 2

FLO-2D Help Resources

2.1 FLO-2D Tutorials

Tutorials

The following tutorials are available on the FLO-2D installation CD and have been loaded on your computer in the FLO-2D Documentation/flo_help/tutorials subdirectory.

There are several workshop lessons located in the FLO-2D Documentation/flo_help/Tutorials/Workshop Lessons subdirectory. The lessons also have review questions that address frequently asked questions.

Lesson 1 – GDS Getting Started

This lesson demonstrates how to create a FLO-2D grid system from a digital terrain model (DTM points) using the Grid Developer System.

Lesson 2 – GDS Spatially Variable Attributes

This lesson describes how to graphically edit the FLO-2D component attributes with the GDS. Part 1 covers general floodplain attributes such as n-values and elevation. Part 2 covers urban features such as streets, levees, ARF and WRF values and infiltration. The tutorial uses a small project with an aerial photo to illustrate editing the component data files.

Lesson 3 – Basic Channel

This lesson provides instruction on creating the CHAN.DAT file from scratch. It facilitates selection of the grid elements for a channel and assignment of channel parameters using aerial photo images as background.

Lesson 4 – HEC-RAS Conversion

In this lesson, you will convert a HEC-RAS cross section data file to FLO-2D XSEC.DAT cross section file format automatically.

Lesson 5 – Profiles

This lesson explains how to interpolate channel slope and geometry using the PROFILES program. It will simplify getting the CHAN.DAT and XSEC.DAT files prepared for the model simulation.

Lesson 6 – MAPPER++

This lesson describes how to view FLO-2D data results in MAPPER++ and how to manipulate the mapping layers. It also covers creating exclusion polygons, hazard maps, damage assessment and layer editing.

Lesson 7 – Rain/Infiltration

This lesson shows how to input global and spatially variable Green-Ampt and SCS curve number infiltration parameters and how to set up and run a simple rain fall simulation using total rainfall and a percent distribution.

Lesson 8 – Natural Channel using HEC-RAS Data

In this lesson the process of loading a HEC-RAS channel and cross section data into a FLO-2D grid system is outlined. It goes over the steps required to create a natural channel from HEC-RAS data.

Lesson 9 – Hydraulic Structures

This lesson outlines the process of creating several hydraulic structures using the GDS.

Lesson 10 – Levees, Walls and Berms

This is an advanced workshop lesson which shows how to create lateral obstructions such as levees and walls using the levee tool in the GDS.

Lesson 11 – Streets and Buildings

For this lesson streets and buildings are created in an urban environment. It uses the polyline street tool and a buildings shapefile.

Lesson 12 – Dam Breach Modeling

This lesson outlines the process for setting up a dam breach model using the NWS Breach embedded model.

Lesson 13 – Multi-Domain Interface (Outflow to Inflow)

This lesson uses the outflow from one grid system as the inflow to another. The lesson describes how to set up outflow nodes that will capture the hydrograph from an upstream system. That outflow data is automatically

assigned to an inflow.dat file for the downstream model. The two models must overlap at the boundary.

Lesson 14 – Advanced Natural Channel

This lesson outlines the complete process for setting up a natural channel. It is a comprehensive channel development guide. This lesson brings together several ideas from Lessons 3, 4, 5 and 8. It also gives the user an guidelines on finalize the channel and prep it for calibration.

2.2 FLO-2D EXAMPLE FLOOD SIMULATIONS

A number of example projects are provided in the FLO-2D Documentation/Example Projects subdirectory. To run the examples, click on the load them in the GDS. You can also run the model from Explorer. First make sure that the FLOPRO.EXE file is in the project subdirectory and double click on the file name. Most of the example simulations are setup for the graphics mode and will generally take only a few minutes to run.

Mudflow Simulation - Barnard Creek, Utah (BARN subdirectory)

An example mudflow simulation is provided for an urbanized alluvial fan (Barnard Creek) near Centerville, Utah. This model simulates a mudflow debouching from a small watershed ravine onto a very steep alluvial fan with numerous streets and buildings. The potential flow surface consists of 521 grid elements (100 ft square). The mudflow enters the grid system at element 486 (a debris basin), flows down a steep street and spreads out into the residential area. The mudflow is viewed overflowing the street, entering side streets and developed lots and becoming more fluid as the floodwave progresses downslope. Buildings have been simulated to account for the loss of storage and flow redirection. The mudflow simulation includes variable sediment concentration and the computation of viscosity and yield stresses. This flood simulation is a good example to review for mudflow, buildings and streets.

Alluvial Fan Channel Overbank Flooding - Monroe Creek, Utah (Monroe folder)

This example flood simulation depicts channel-floodplain discharge exchange on a large, undeveloped alluvial fan. The grid system consists of 3,026 grid elements, 200 ft on a side. The flood hydrograph has a peak discharge of 1,900 cfs. The channel is represented by 47 channel elements with variable channel geometries; rectangular, trapezoidal and natural-shaped. The flow is confined along the channel by berms that have been represented by higher grid element elevations. The overbank flooding can be observed returning to the channel. This simulation is a good example for reviewing the variable cross section data in the CHAN.DAT file.

Desert Southwest Alluvial Fan Flooding - Whiskey Pete's, Nevada (Whiskey folder)

Whiskey Pete's is a classic conically-shaped desert alluvial fan. Multiple distributary channels and infiltration are simulated. The hydrograph peak discharge is in excess of 13,000 cfs. There is no development or channelization. This flood simulation is a good example for reviewing the multiple channel option.

Rainfall/Runoff - Blue Diamond Watershed, Nevada (Diamond subdirectory)

Hint:

Review the Example Project data files to start your own project. Try to find an example that is similar to the project you are working on.

Rainfall, infiltration and runoff are simulated for a 34 square mile desert watershed with 10,472 grid elements 300 ft square. Steep, unvegetated ridges, bedrock and numerous small watershed ravines characterize the project area. The drainage pattern can be observed as the rainfall/runoff develops in the graphics mode. The higher elevations and ridges drain first and the water collects in the various ravines and gullies. This simulation represents a good example for reviewing rainfall and infiltration.

Large River Flood Inundation - Truckee River, Reno Nevada (Truckee subdirectory)

In January 1997, the Truckee River flooded a large area including portions of the cities of Reno and Sparks, Nevada. FLO-2D was used to replicate this flood. Over 100 channel cross sections were surveyed to simulate the overbank flooding. Bridges, some streets and buildings, infiltration and levees are simulated. Partial bridge blockage is simulated with a doubled pier width. The flood inundation covers a large, flat area that requires a long time to drain. There are three tributaries in this reach, each with an inflow hydrograph. This example project has a number of interesting components including bridges, levees, streets and channel confluences.

Urban Flooding at the Ocean – Waikiki Beach, Oahu, Hawaii (Alawai subdirectory)

This flood example simulation includes a number of components associated with urban flooding such as street flow and numerous buildings. Alawai Canal runs through the center of the project area and is open to the ocean. Excess rainfall runs off steep watersheds and enters channels that join the canal system in Waikiki beach. The Alawai Canal is nearly flat and is filled with water from the ocean at the start of the simulation.

Ocean Storm Surge/Tsunami Model in an Urban Area – Waikiki Beach, Oahu, Hawaii (Alawai-Tsunami subdirectory)

By assigning stage-time relationships to the outflow elements along the Waikiki coast line, the Alawai watershed model is converted to an ocean storm surge or tsunami model. A high water surface elevation is specified for the coastal elements for a short duration. This results in a rapid progression of

the ocean storm surge over the urban area. The ocean surge enters streets and the Alawai Canal in the center of the city. The model demonstrates the application of the FLO-2D model to simulate the overland progression of hurricane storm surges or tsunami waves in urban areas.

Large River Flooding – Rio Grande, New Mexico (Rio Grande subdirectory)

Over 173 miles of the Middle Rio Grande is simulated using actual cross section data. This example represents a typical river-floodplain exchange of discharge. The river floodplain is confined by levees along most of its length. Use this flood simulation to review the data input in the XSEC. DAT and CHAN.DAT files.

Working with Geo-referenced Images – Goat Camp Creek, Gila County, Arizona (Goat subdirectory)

This small project provides an opportunity to work with the GDS editor components and capabilities. The aerial photos can be imported and used to edit the various model components such as channels, streets, ARF's and WRF's, and levees. This flood simulation includes channel overbank flow from a small river through an urban area.

Hint:

Copy the following Processor Programs into each project folder. Double click the executable name to run the program:

- FLO.EXE
- PROFILES.EXE
- MAXPLOT.EXE
- HYDROG.EXE

2.3 FLO-2D POWERPOINT PRESENTATIONS

A series of PowerPoint presentations are available in the FLO-2D help folder. These presentations offer guidance on modeling many of FLO-2D components. The files can be found in FLO-2D Documentation\flo_help\PowerPoint Presentations.

TABLE 2.1. ENGLISH/METRIC

Variable	English	Metric
discharge	cfs	m^3/s (cms)
depth	ft	m
hydraulic conductivity	inches/hr	mm/hr
rainfall and abstraction	inches	mm
soil suction	inches	mm
velocity	fps	mps
volume	acre-ft	m^3 (cm)
viscosity	poise (dynes-s/cm^2)	poise

2.4 OTHER HELP DOCUMENTS

Several documents files are provided in the FLO-2D Handout folder. These files provide advanced model guidance and discussion on specific components, porting data files between versions, mudflows and other topics. The files can be found in FLO-2D Documentation\flo_help\Handouts.

2.5 METRIC OPTION

The user can choose either the English or Metric system of units (for the Metric system set METRIC = 1 in the CONT.DAT file). When using the Metric system, substitute the appropriate metric unit for the English unit in the data files. The following basic units are used in the model:

CHAPTER 3

PRE-PROCESSOR PROGRAMS

3.1 INTRODUCTION

There are two pre-processor programs to help to create or edit the FLO-2D data files: GDS and PROFILES. Tutorials and workshop lessons for some of the programs' functions are available in the FLO-2D/flo_help subdirectory. A discussion of the commands in the PROFILES program is included in this manual. The description of the GDS functions and commands are presented in a separate manual.

3.2 GDS

The grid developer system (GDS) will graphically create and edit the FLO-2D grid system and its attributes. It generates a grid system from a digital terrain map or a set of DTM points and interpolates and assigns elevations to the grid elements. It will also create the basic FLO-2D data files for an overland flow flood simulation. Grid element attributes such as channels, streets, levees and n-values can be developed graphically. It is recommended that you use the GDS for all data file development and graphical editing. The GDS has a separate reference manual. In addition, there are a number of GDS tutorials and workshop lessons that are available in the FLO-2D Documentation\flo_help subdirectory.

3.3 PROFILES

The PROFILES processor program displays the channel slope and permits interactive adjustment of the channel bed elevation, channel depth, channel n-values and channel geometry. It will display the channel cross section geometry and

interpolate the slope and cross section geometry between surveyed cross sections. PROFILES can also be used to view output water surface profiles (see the Post-Processor Programs Section). To run PROFILES, access the program from the GDS File menu or copy the program to the project subdirectory and double click the name in an browser window.

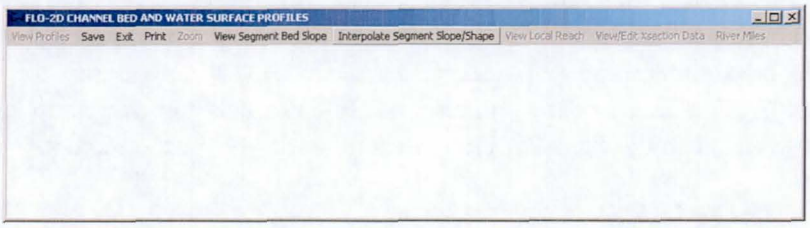
Before using the PROFILES program, the basic FLO-2D files plus the CHAN.DAT file have to be created. The XSEC.DAT will also have to be created if surveyed cross section data will be used. The general procedure for using the PROFILES program is as follows:

> Hint:
> **To quickly access the PROFILES program, copy the executable PROFILES.EXE into the project subdirectory and double click on it.**

1. Create the six basic FLO-2D data files.
2. Develop the XSEC.DAT file for surveyed cross section data if necessary.
3. Complete the channel data file (CHAN.DAT) based on rectangular, trapezoidal or surveyed (natural) cross sections.
4. For surveyed cross sections, identify the channel element cross section number XSECNUM in the CHAN.DAT file to represent the cross section. All other XSECNUM's will be assigned a zero '0' value.
5. Run the PROFILES program from the GDS or Explorer.
6. The model bed slope can be compared with surveyed bed elevations by developing the WSURF.DAT file.
7. Upon exiting the PROFILES program, two new files FPLAIN.NEW and CHAN.NEW will be created that contain the adjusted bed elevations and channel depths. You will be prompted to save your data by replacing the original FPLAIN.DAT and CHAN.DAT files with the newly created FPLAIN.NEW and CHAN.NEW files. If you choose not to replace the files, the two new files will still be available for review.

> Hint:
> **When assigning cross section numbers to the CHAN.DAT file, make sure all channel elements that are not associated with a surveyed cross section have XSECNUM values of zero**

Initially the PROFILES program will display a blank screen with a Main Menu showing options to 'View Segment Bed Slope' or 'Interpolate Segment Slope/Shape'.



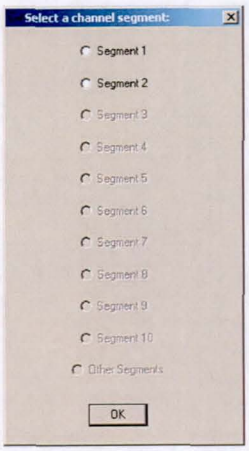
Interpolating a New Channel with Surveyed Cross Sections

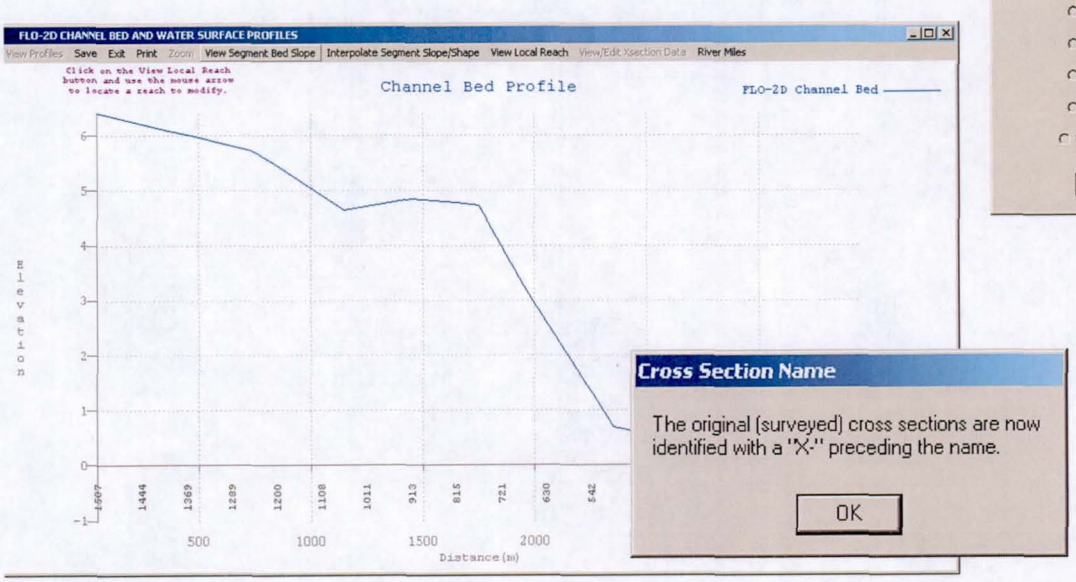
To interpolate the cross sections and slope and assign a cross section to every channel element in PROFILES, use the 'Interpolate Segment Slope/Shape' menu option as follows:

1. Select a channel segment from the list provided in the dialog box shown in the sidebar. If there is only one channel segment, the interpolation will be completed directly. Note that before interpolation, the channel slope profile may look like a stair case, this is because only the surveyed cross sections define the channel profile at this point. Following interpolation the slope profile will be more representative of the actual river profile.

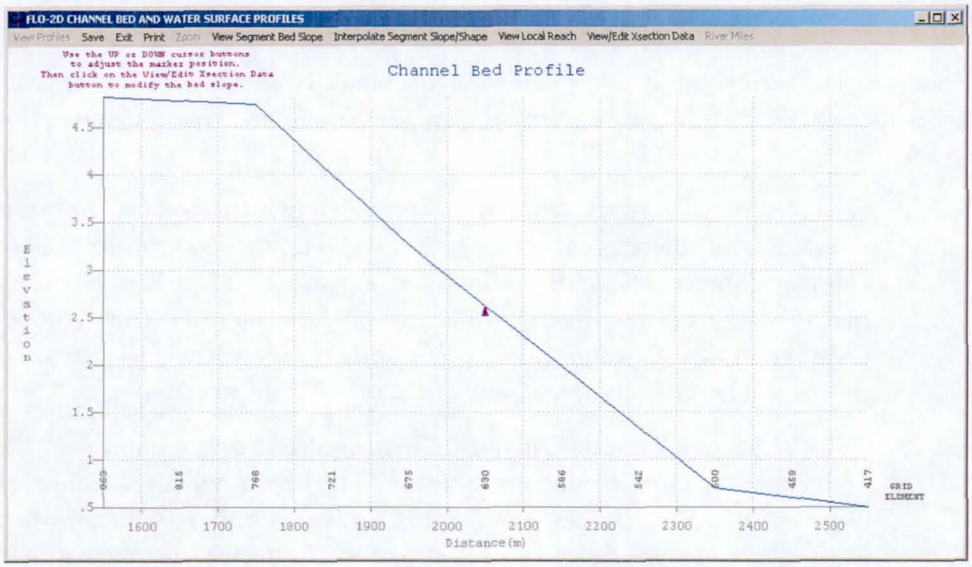
2. PROFILES will automatically locate the surveyed cross section data and interpolate the cross section geometry and elevation (thalweg slope) for all those channel elements between the surveyed cross sections within the segment. The following dialog box will appear indicating that the original cross sections have been renamed with a prefix 'X-' before each cross section name.

3. Click 'OK' in the dialog box to view the interpolated bedslope.



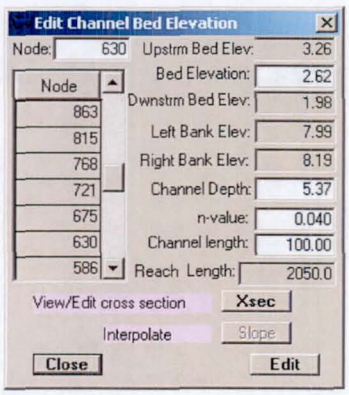


4. Click on the 'View Local Reach' button on the menu bar. Click anywhere along the bedslope profile to zoom in on a local reach of 10 channel elements.

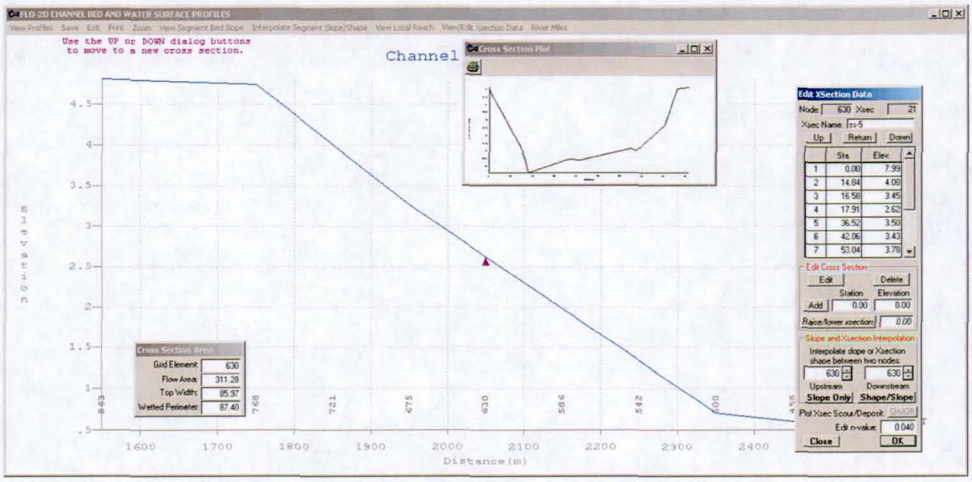


Note:
Refer to the Workshop Lesson PROFILES tutorial for a more detailed example of this procedure.

5. Click on 'View/Edit Cross Section Data' to view the following dialog box displaying the channel element characteristics:



6. Click on the 'Xsec' button in the dialog box to view the cross section data and image.



Hint:

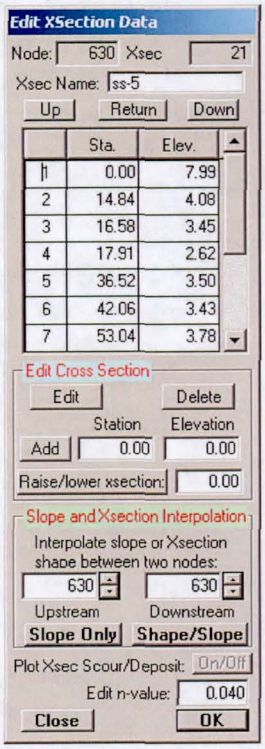
When interpolating between two cross sections, they must be entered in order from upstream to downstream..

7. At this point you may view some of the other cross sections by clicking on the "Up" and "Down" buttons in the dialog box. The computed cross section geometry and all the cross section station and elevation data can be reviewed and edited. You can edit the channel and cross section data by adding or deleting stations and elevations, revising the Manning's n-value, or raising or lowering the entire cross section.

8. You can now further interpolate both the bedslope and the channel geometry. Simply identify the Upstream and Downstream channel elements in the group boxes labeled 'Slope and Xsection Interpolation'. You can use the 'Up' and 'Down' buttons to locate one of the surveyed cross sections and then type in the other either upstream or downstream channel elements. There may be several channel elements between two cross sections selected for interpolation. Click on either the 'Slope Only' or 'Shape/Slope' buttons to interpolate either the channel bed slope or slope and the cross section shape. It may be necessary to interpolate only those cross sections that you have edited, but you need to interpolate both upstream and downstream of revised cross section. The cross section geometry is linearly interpolated according to top width and distance and is adjusted by a weighted flow area. Essentially one cross section is overlaid on the other cross section, stretched or contracted and the elevations averaged.

9. As you work, save the results frequently by clicking on 'Save' on the menu bar.

NOTE: You can also perform the initial channel interpolation of the cross sections automatically in the GDS.



INPUT DATA FILE DESCRIPTION

4.1 GENERAL

The FLO-2D data file variables and format are described in this chapter. These files are called directly by the model during a flood simulation. For each data file a list of the variables, a portion of an example data file, and an alphabetical description of the variables are presented. Some instructional comments follow the descriptions to clarify the variable's application. TheGDS or any ASCII text editor can be used to create or edit the data files. Also use the GDS to create and graphically edit the data files.

All of the data entries are in free format requiring only a space to separate the variables. All ID characters (letters) are case sensitive. The variables are listed in this manual line by line. Each line may contain several variables that are highlighted by bold text and capital letters. Array variables are indexed as shown in the following example from the INFLOW.DAT file, Line 3:

Line 3 **HYDCHAR = 'H', HP(I,J,1), HP(I,J,2), HP(I,J,3)**
I = 1, J = 1, Number of inflow hydrograph pairs

where:

I and J are array indices representing grid element number adn hydrograph pair;
H(I,J,1) is the time (hrs) at the start of each discretized interval in the channel inflow hydrograph;
H(I,J,2) is the discharge (cfs);
H(I,J,3) is the sediment concentration by volume for a mudflow inflow hydrograph.

HDYCHAR is a line identifier character 'H'.

The variables in Line 3 on the INFLOW.DAT file represent one line of a discretized inflow hydrograph that is repeated for each of the hydrograph pairs for each inflow grid element. The data represented by Line 3 for the first four time steps is as follows:

INFLOW.DAT Variable Example

HP(J,1)	**HP(J,2)**	**HP(J,3)**
Time	Discharge	Sediment Conc..
(hrs)	(cfs)	% by Volume
0.0	0.0	0.00
0.1	10.0	0.00
0.2	25.0	0.20
0.3	50.0	0.25

Backup files of the data files (*.BAC) can be created when program reads the data. The backup option is invoked by a switch (IBACKUP) in the CONT.DAT file. The backup files are useful to locate data errors.

4.2 LIST OF PROGRAM FILE UNITS

The following table lists the data and output file ('Unit') numbers that are assigned by the FLO-2D model at runtime. These unit numbers may be reported in error messages accompanying a premature simulation termination. Referring to these numbers may help you to debug input data errors.

TABLE 4.1. LIST OF FLO-2D FILES AND UNIT NUMBERS

File No.	File Name	File No.	File Name	File No.	File Name	File No.	File Name
2	BASE.OUT	55	SURFAREA.OUT	105	MAXQHYD.OUT	254	USED
7	ERROR.CHK	56	CHANNEL.CHK	106	MAXQBYDIR.OUT	255	BREACH.OUT
8	Used	57	LEVEE.DAT	107	MAXQRESOLVED.OUT	260	BREACH.BAC
9	TOLER.DAT	58	LEVEE.BAC	108	FPLAIN.RGH	261	DEPCHFINAL
10	CADPTS.DAT	59	LEVEE.OUT	109	CHAN.RGH	262	VELCHFINAL
11	SUMMARY.OUT	60	DEPFP.OUT	110	STREET.RGH	289	INFIL_DEPTH.OUT
12	Used	61	DEPCH.OUT	111	FPINFILTRATION.OUT	300	BATCHCYCLE.DAT
13	Used	62	FINALDEP.OUT	112	Used	321	REVISED_RATING_TABLE.OUT
14	CHANMAX.OUT	63	VELDIREC.OUT	117	VELTIMEC.OUT	311	MANNINGS_N.DAT
15	Used	64	VELFP.OUT	118	VELTIMFP.OUT	333	NOSHOW.CHK
16	Used	65	SUPER.OUT	119	CHANBANK.DAT	397	TOPO.BAC
17	SEDFP.OUT	66	CHVOLUME.OUT	120	FPXSEC.DAT	398	MANNINGS_N.BAC
18	SEDCHAN.OUT	67	INFILHY.OUT	121	FPXSEC.BAC	528	DEPTHDUR2.OUT
19	TIME.OUT	68	HYDROSTRUC.DAT	122	Used	1557	SWMMFLO.DAT
20	DEPTH.OUT	69	HYDROSTRUC.BAC	123	HYSTREET.OUT	1558	SWMM.RAIN
21	TIMDEP.OUT	70	HYDROSTRUCT.OUT	124	MAXWSELEV.OUT	1559	SWMMFLORT.DAT
22	ROUGH.OUT	71	Used	125	FPFROUDE.DAT	1560	SWMMQIN.OUT
23	STREET.OUT	72	STVELDIR.OUT	128	VELTIMEST.OUT	1561	Used
24	VELOC.OUT	73	STVEL.OUT	139	FPSEDSIZE.OUT	1562	SWMMOUTF.DAT
25	MULTCHN.OUT	74	CHNBEDEL.OUT	140	CHANSEDSIZE.OUT	1563	FPRIMELEV.OUT
26	FPREV.NEW	75	STRELEV.OUT	150	VOLUMEBINARY.OUT	1564	SWMMOUTF.BAC
27	DEPTHTOL.OUT	76	Used	151	FPLAINBINARY.OUT	1565	SWMMOUTFIN.OUT
28	DEPTHDUR.OUT	77	SEDCONSERV.OUT	152	CHANBINARY.OUT	1566	Used
29	TOLER.BAC	78	Not used	153	STREETBINARY.OUT	1567	SWMMFLO.BAC
30	CONT.DAT	79	CROSSQ.OUT	154	CROSSBINARY.OUT		
31	FPLAIN.DAT	80	HYCHAN.OUT	155	HYSTRUCBINARY.OUT		
32	RAIN.DAT	81	FINALVEL.OUT	156	XSECSEDBINARY.OUT		
33	INFIL.DAT	82	Used	157	SEDBINARY.OUT		
34	INFLOW.DAT	83	FINALDIR.OUT	159	Used		
35	EVAPOR.BAC	85	XSEC.DAT	160	LEVOVERTOP.OUT		
36	CHAN.DAT	86	CHANBANKEL.CHK	161	LEVEEDEFIC.OUT		
37	ARF.DAT	87	OVERBANK.OUT	162-170	CADPTS_DSx.DAT x = 1,9		
38	MULT.DAT	88	SEDTRAN.OUT	171-179	INFLOWx_DS.DAT x = 1,9		
39	SED.DAT	89	RAINCELL.DAT	169	Used		
40	CONT.BAC	90	CHANSTABILITY.OUT	180	WSTIME.DAT		
41	FPLAIN.BAC	91	XSECAREA.OUT	181	WSTIME.OUT		
42	RAIN.BAC	92	INTERGWS.OUT	190	TIMETWFT.OUT		
43	INFIL.BAC	93	CHANWS.OUT	191	TIMETOPEAK.OUT		
44	INFLOW.BAC	94	XSEC.OUT	192	TIMEONEFT.OUT		
45	FLOODWAY.OUT	95	EVAPOR.DAT	193	FLOODTIME.OUT		
46	CHAN.BAC	96	HYCROSS.OUT	200	Used		
47	ARF.BAC	97	TOPO.DAT	201	Used		
48	MULT.BAC	98	Used	202	Used		
49	SED.BAC	99	Used	203	Used		
50	OUTFLOW.DAT	100	Used	204	Used		
51	OUTFLOW.BAC	101	IMPACT.OUT	205	MULTSTEEP.OUT		
52	STREET.DAT	102	STATICPRESS.OUT	208	MULT.RGH		
53	STREET.BAC	103	CONFLUENCE.OUT	226	FPFROUDE.BAC		
54	CROSSMAX.OUT	104	SPECENERGY.OUT	250	BREACH.DAT		

4.3 DATA FILES

Four data files are required for every flood simulation: CONT.DAT, TOLER. DAT, FPLAIN.DAT, CADPTS.DAT. The INFLOW.DAT and OUTFLOW.DAT files are optional, but typcially are necessary in a FLO-2D flood simulation. The CADPTS.DAT is not listed. Although, it is required for every flood simulation, this file is automatically created by the GDS and is complete. It will not need editing unless you add or delete grid elements to model after the data files are established. The TOPO.DAT and MANNINGS_N.DAT files are generated by the GDS and the FLO-2D Pro model at runtime. These files can be used for GIS and CADD applications, but are not required.

FILE: CONT.DAT

SYSTEM CONTROL DATA

CONT.DAT File Variables

24.0 0.10 2 0 0	Line 1:	**SIMULT TOUT LGPLOT METRIC IBACKUP**
1 1 0 0 0	Line 2:	**ICHANNEL MSTREET LEVEE IWRFS IMULTC**
0 1 0 0 0 0 0	Line 3:	**IRAIN INFIL IEVAP MUD ISED IMODFLOW SWMM**
0 0 0	Line 4:	**IHYDRSTRUCT IFLOODWAY IDEBRV**
0.000 0.0 0.0 0.30 0.70 0.150	Line 5:	**AMANN DEPTHDUR XCONC XARF FROUDL SHALLOWN ENCROACH**
2 0	Line 6:	**NOPRTFP SUPER**
2	Line 7:	**NOPRTC**
0 0.0	Line 8:	**ITIMTEP TIMTEP**
0.1	Line 9:	**GRAPTIM**

Notes:
Line 5: If IFLOODWAY = 0 omit ENCROACH
Line 7: If ICHANNEL = 0 omit Line 7
Line 9: If LPLOT = 0 omit Line 9

CONT.DAT File Example

```
24.0  0.10  2  0  0
1  1  0  0  0
0  1  0  0  0  0  0
0.000  0.0  0.0  0.30  0.70  0.150
2 0
2
0  0.
0.1
```

Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
AMANN	r	-0.4 to 0.4 -99 > 1.0	Increments the floodplain Manning's n roughness coefficient at runtime. AMANN will uniformly increase or decrease the every floodplain grid element n-value (n-value + AMANN). Set AMANN to a negative value to decrease the n-value. Set AMANN= -99 to turn off depth integrated roughness (see comment 9). Set AMANN > 1.0 to globally increase the n-value multiplicatively (n-value * AMANN).
DEPTHDUR	r	0.01 - 100 0.003 - 30	Flow depth (ft or m) for a depth-duration analysis. When a flow depth greater than DEPTHDUR is computed, the time duration of inundation for that grid element is tracked and reported in the DEPTHDUR.OUT file (see comment 8).
ENCROACH	r	0 - 10 0 - 3	The floodway encroachment increase in flow depth (ft or m). The IFLOODWAY switch must be set to 1 and a previous FLO-2D simulation must be completed for the project to generate the maximum water surface elevations.
FROUDL	r	0 - 5	Maximum Froude number for overland flow. When FROUDL is exceeded, the floodplain n-value is increased by 0.001 for that grid element for the next timestep (see comment 3). Set FROUDL = 0.0 for no Froude number limitations. The increased n-values are reported in the ROUGH.OUT and FPLAIN.RGH files (see comments 3 and 4).
GRAPTIM	r	0.01 - ∞	Time interval in hours that the graphics display is updated (e.g. set GRAPTIM = 0.02 for a frequent update). GRAPHTIM is required when LGPLOT = 2. This variable will not affect the file output data time interval (TOUT). The graphics mode is limited to a 48-day inflow hydrograph.
IBACKUP	s	0 = off 1 = on 2	IBACKUP = 1 creates a backup file of all the data files with the *.BAC extension for data error troubleshooting. It also enables the model to be resumed following termination from the last output interval. IBACKUP = 2 enables elevation changes for outflow nodes made at runtime to be permanently written to the FPLAIN.RGH file (see comment 10).

Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ICHANNEL	s	0 = off 1 = on	If ICHANNEL = 1 main channel flow will be simulated. The CHAN.DAT must be created (see comments 1 and 6).
IDEBRV	s	0 = off 1 = on	Set IDEBRV = 1 if a debris basin volume should be filled before routing the flow hydrograph.
IEVAP	s	0 = off 1 = on	Set IEVAP = 1 if simulating free water surface evaporation from overland flow or channel flow.
IFLOODWAY	s	0 = off 1 = on	If FLOODWAY =1 a floodway analysis will be performed in the next FLO-2D simulation. An initial FLO-2D flood simulation must be completed prior to a floodway simulation (see comment 5).
IHYDRSTRUCT	s	0 = off 1 = on	Set IHYDRSTRUCT = 1 to simulate hydraulic structures either on the floodplain or in the channel. The HYSTRUC. DAT file must be created.
IMULTC	s	0 = off 1 = on	Set IMULTC = 1 to simulate multiple channel (rill and gully flow) on the floodplain. Rill and gully flow rather than overland sheet flow will be simulated to route the flow between designated floodplain grid elements. The MULT.DAT file must be created.
IMODFLOW	s	0 = off 1 = on	Set IMODFLOW = 1 to simulate surface-groundwater interaction. This switch initiate the MODFLOW groundwater model a during the FLO-2D simulation.
INFIL	s	0 = off 1 = on	INFIL = 1 initiates an infiltration subroutine using the Green-Ampt infiltration model for either channel or overland infiltration. The INFIL.DAT file must be created.
IRAIN	s	0 = off 1 = on	Set IRAIN = 1 to simulate rain on the grid system. The RAIN.DAT file must be created (see comment 1).
ISED	s	0 = off 1 = on	If ISED = 1, the sediment transport routine will be used. The SED.DAT file must be created.
ITIMTEP	s	0 = off 1 = on	Set ITIMTEP = 1 to create file of time lapse flow depths for a MAXPLOT or MAPPER post-simulation flood animation. The timestep TIMTEP must be specified.

Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
IWRFS	s	0 = off 1 = on	IWRFS = 1 specifies that area and width reduction factors (ARFs and WRFs) will be assigned in the ARF.DAT file.
LEVEE	s	0 = off 1 = on	Set LEVEE = 1 to simulate levees. The LEVEE.DAT file must be created.
LGPLOT	s	0 = text 2 = graphic	LGPLOT = 0 will display a text message on the screen which scrolls the simulation time, minimum timestep and volume conservation. LGPLOT = 2 provides a graphic display of the floodwave progression over the grid system (flow depth) and inflow hydrograph.
METRIC	s	0 = Eng 1 = Met	METRIC = 0 for English units and METRIC = 1 for the metric system of units.
MSTREET	s	0 = off 1 = on	MSTREET = 1 to initiate the street flow component. The STREET.DAT file must be created (see comment 2).
MUD	s	0 = off 1 = on	Set MUD = 0 for clear water and MUD = 1 for hyperconcentrated sediment flow. Setting MUD = 1 requires a sediment concentration by volume. Either the variable HP(I,J,3) for a floodplain hydrograph or H(I,J,3) for a channel hydrograph to be assigned to each inflow hydrograph interval (see comments 1 and 3). The SED.DAT file must be created.
NOPRTC	s	0, 1 or 2	If NOPRTC = 0, all the channel data is printed to the BASE.OUT file. If NOPRTC = 1, the channel outflow data is not printed to the BASE.OUT file. If NOPRTC = 2, all the channel output data is not printed..
NOPRTFP	s	0, 1, 2 or 3	If NOPRTFP = 0, all the floodplain flow data is printed to the BASE.OUT file. If NOPRTFP = 1, floodplain outflow data is not printed to the BASE.OUT file. If NOPRTFP = 2, the floodplain flow data is not printed to the BASE.OUT file. If NOPRTFP = 3, only floodplain outflow data is printed to the BASE.OUT file.

Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
SHALLOWN	r	0 - 0.4	Flow roughness n-value for shallow overland flow (flow depth < 0.2 ft or 0.6 m) (see comment 9). If SHALLOWN < 0.05, the model will default to SHALLOWN = 0.100.
SIMUL	r	0.01 - ∞	Simulation time (hours).
SUPER	s	0 = off 1 = on	SUPER = 1 prints a message that the flow is supercritical in the given grid element to the SUPER.OUT file. IF SUPER = 0, the message will be not be printed for either overland and channel flow.
SWMM	i	0 = off 1 = on	SWMM = 1 initiates the EPA SWMM storm drain model interface with the FLO-2D predicted surface water.
TIMTEP	r	0 - 100	An output time interval (hrs) that the flow depth will be written to an output file for a post-simulation flood animation in MAXPLOT or MAPPER. TIMTEP should be a multiple of TOUT. The variable ITIMTEP must be set.
TOUT	r	0.01 - ∞	Output time interval (hours). This is the time interval that hydraulic data is recorded in the various output files *.OUT.
XARF	r	0 - 1	Global area reduction factor applied to all grid elements. This factor reduces the grid element surface area available for flood volume storage. XARF can be used to account irregular surface topography, dense vegetation or other features. Range: 0 < XARF < 1. A typical value for XARF might be 0.10 indicating that 10% of each grid element surface is not available for flood storage. The XARF value is overridden by the ARF variables specified for the individual grid elements in the ARF.DAT file. Assign XARF = 0 to flood the entire surface area of the grid elements.
XCONC	r	0 - 0.50	Volumetric concentration to bulk the inflow discharge hydrograph (channel or floodplain). For example, set XCONC = 0.20 for a concentration of 20% by volume. This will account for sediment bulking without initiating the hyper-concentrated sediment transport routine. If simulating clear water flooding, set XCONC = 0. Set MUD = 0, if XCONC is greater than zero.

Instructional Comments

CONT.DAT File

The following comments will assist in the developing of the CONT.DAT file:

1. If any of the switches MUD, ISED, IRAIN, IMULT, INFIL, MSTREET, LEVEE, ICHANNEL, IWRFS, IMODFLOW, SWMM or IHYDRSTRUCT are set to 0 "off", then the corresponding data file can be omitted. For example, set MSTREET = 0 and the STREET.DAT file can be omitted.

2. Streets, groundwater, mudflow, levees, and rill and gully flow can be simulated with or without a channel.

3. On alluvial surfaces, supercritical flow is uncommon and may be inhibited by sediment entrainment. There are three possible approaches to a high Froude number flow analysis

   a. Allow supercritical flow and do not limit the Froude number.

   b. Increase the grid element roughness by assigning AMANN or setting higher individual grid element n-values to reduce the Froude number (assign spatially variable n-values).

   c. Limit the Froude number to a specific value (e.g. set FROUDL = 0.99 or 1.11). When FROUDL is exceeded the grid element roughness value will be increased by 0.001 for the next timestep. After simulating a flood, review the ROUGH.OUT file to determine where FROUDL was exceeded and the maximum n-values that were computed. You may consider revising the n-values in the FPLAIN.DAT file to match those in the ROUGH.OUT file if you want to ensure that FROUDL is not exceeded. You can do this automatically by renaming the FPLAIN.RGH file to FPLAIN.DAT

4. The floodwave travel time should be reviewed to determine if it is appropriate. The travel time can also be used to calibrate the n-values. Adjusting n-values with FROUDL will slow the arrival of the frontal wave. During the hydrograph recessional limb when the Froude number is less than 0.5 and the flow is shallow, the n-value decreases by 0.0005 until the original n-value is reached

5. IFLOODWAY initiates the floodway routine. Flow will not be exchanged between floodplain grid elements unless the maximum water surface plus the encroachment depth (ENCROACH) from a previous FLO-2D simulation is exceeded. See the Floodway discussion in the Reference Manual component descriptions. An initial FLO-2D simulation is required to establish the maximum water surface elevations.

6. If channel flow is simulated (ICHANNEL = 1), then the NOPRTC variable must be set in CONT.DAT. In addition, any channel outflow control must be defined in OUTFLOW.DAT.

7. ITIMTEP will enable a simple animation of the overland flow to be shown in Mapper or MAXPLOT. The animation will be based on a time interval TIMTEP specified by the user.

8. The depth duration analysis is used to determine how long a floodplain grid element is inundated at a flow depth greater than the DEPTHDUR variable. If DEPTHDUR = 1 ft, the output file DEPTHDUR.OUT has the total duration in hours that the depth exceeded 1 ft. The results can be reviewed in MAXPLOT. If the depth duration analysis is activated, then a second output file DEPTHDURATION2.OUT is generated for the cumulative time duration above 2 ft (0.61 m).

9. To improve the timing of the floodwave progression through the grid system, a depth variable roughness can be assigned. The basic equation for the grid element roughness n_d as function of flow depth is:

$$n_d = n_b * 1.5 * e^{-(0.4\ \mathrm{depth}/\mathrm{dmax})}$$

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = flow depth for drowning the roughness elements and vegetation (hardwired 3 ft or 1 m)

This equation prescribes that the variable depth floodplain roughness is equal to the assigned flow roughness for complete submergence of all roughness (assumed to be 3 ft or 1 m). This equation is applied by the model as default and the user can 'off' the depth roughness adjustment coefficient for all grid elements by assigning AMANN = -99. This roughness adjustment will slow the progression of the floodwave. It is applied valid for flow depths ranging from 0.5 ft (0.15 m) to 3 ft (1 m). For example, at 1 ft (0.3 m), the computed roughness will be 1.31 times the assigned roughness for a flow depth of 3 ft. Assigning a ROUGHADJ value may reduce expected high Froude numbers. When using the combined shallow roughness value and the depth variable roughness, the following rules apply:

0.0 < flow depth < 0.2 ft (0.06 m)	n = SHALLOWN value
0.2 ft (0.06 m) < flow depth < 0.5 ft (0.15 m)	n = SHALLOWN/2.
0.5 ft (0.15 m) < flow depth < 3 ft (1 m)	$n = n_b * 1.5 * e^{-(0.4\ \mathrm{depth}/\mathrm{dmax})}$
3 ft (1 m) < flow depth	n = n-value in FPLAIN.DAT

10. IBACKUP = 1 is used to create a backup file with an *.BAC extension. This file can be reviewed to see if the model is correctly reading the data. This is a format troubleshooting tool. These files can be renamed to *.DAT and the model will run with them.

    It will also be used to generate a series of binary files that represent the model results at the last output interval. The binary files are overwritten at the end of each output interval so if the model is terminated prior to the end of the run for any reason, the simulation can be restarted from the last interval. Setting the switch to 1 can significantly lengthen the model run time.

    Setting IBACKUP = 2 will write all elevation changes associated with the outflow nodes and channel top-of-bank revisions to the FPLAIN.RGH file. Outflow nodes used by the model will discharge water off the grid system using an upstream normal depth flow approximation. They must be lower than the upstream contiguous elements and at runtime the model will automatically adjust the grid element elevation 0.25 ft lower than the lowest upstream grid elements. If the channel top-of-bank elevation is different from the floodplain bank elevation by more than 1 ft (0.3 m), the floodplain grid element is revised to the top-of-bank elevation. These revisions are written to the FPLAIN.RGH file and the file can be renamed to FPLAIN.DAT to run the model.

FILE: TOLER.DAT

NUMERICAL STABILITY CONTROL DATA

TOLER.DAT File Variables

0.1 0.20 1.0	Line 1: **TOL DEPTOL WAVEMAX**
C 0.6 0.5 0.5	**Line 2: COURCHAR = 'C' COURANTFP COURANTC COURANTST**
T 0.1	**Line 3: COURCHAR = "T" TIME_ACCEL**

TOLER.DAT File Example

```
0.1 0.20 1.0
C 0.6 0.5 0.5
T 0.1
```

Variable Descriptions for the TOLER.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
COURANTC	i	0.2 - 1 typ 0.3 - 0.7	Courant number for channels. Courant-Friedrich-Lewy numerical stability parameter that relates the floodwave movement to the discretized routing model.
COURANTFP	i	0 - 1 typ 0.3 - 0.7	Courant number for floodplain. Courant-Friedrich-Lewy numerical stability parameter that relates the floodwave movement to the discretized routing model. (See comment 4)
COURANTST	i	0 - 1 typ 0.3 - 0.7	Courant number for streets. Courant-Friedrich-Lewy numerical stability parameter that relates the floodwave movement to the discretized routing model.
COURCHAR	c	C, T	Character 'C' that identifies Line 2 with the Courant stability parameter.
DEPTOL	r	0 or 0.1 - 0.5	Tolerance value for the percent change in the channel flow depth for a given timestep. When DEPTOL is exceeded for any channel element, the timestep will be reduced. If DEPTOL = 0, then the timestep is governed by the prescribed numerical stability criteria (see comment 3). Suggested default value for DEPTOL = 0.0
TIME_ACCEL	r	0.01 - 100.00	Coefficent to increase the rate of incremental timestep change. Default value - 0.1. Higher value may result in a faster simulation time.
TOL	r	0 - 0.5	Surface detention. TOL is a minimum value of the flow depth for flood routing. A typical value is 0.10 ft (see comment 2). Use a small TOL value for rainfall runoff (0.025 ft to 0.05 ft; 0.01 m to 0.015 m). A TOL value for streets (0.03 ft or 0.01 m) is hardwired in the model.
WAVEMAX	r	0.00 - 2.0	Maximum value of the numerical stability coefficient for full dynamic wave flood routing. Default value for WAVEMAX = 0.0. WAVEMAX directly controls the maximum timestep. Reduce WAVEMAX to improve volume conservation or reduce surging. Increase WAVEMAX to speed up the flood simulation.

Instructional Comments for the TOLER.DAT File

Set the Courant numbers to 0.6 (default), DEPTOL = 0. and WAVEMAX = 0 for the intial simulations. If numerical surging is observed, first reduce the Courant numbers, then add DEPTOL = 0.2 and WAVEMAX as follows:

OPTION	RESULT
1. Set WAVEMAX in the range 0.10 to 1.00 (typical value = 0.25).	Dynamic wave stability criteria increments and decrements the computational timestep when WAVEMAX is exceeded. Model runs more slowly but is stable.
2. Assign WAVEMAX as a negative number using same range of values -0.10 to -1.00 (typical value = -0.25).	The dynamic wave numerical stability criteria are turned off. The timestep will be incremented of decremented only by Courant stability criteria or the DEPTOL. The floodplain Manning's n-values are incremented when the dynamic wave stability criteria is exceeded, but the timestep is not decreased. The model runs faster and makes n-value adjustment in grid elements with potential stability problems. Changes to n-values are written to the ROUGH.OUT file. The increase and decrease of the n-values occurs according to the following relationships: $n = n + 0.0006616\ e^{(-10.9\ n)}$ when the limiting timestep is exceeded $n = n + 0.00005$ when the limiting timestep is not exceeded.
3. WAVEMAX = 0 or > 100 (typical value 100.25).	The dynamic numerical stability criteria are turned off. The timestep are incremented or decremented only by the Courant stability criteria and the DEPTOL value. There is no n-value adjustment.

1. The TOL value prescribes the flow depth for a floodplain or channel grid element below which no flood routing will be performed. TOL is analogous to a depression storage rainfall abstraction. The TOL value for streets is hardwired (0.03 ft or 0.01 m).

2. DEPTOL controls the percent change in channel flow depth for a given timestep. It is a generic control that eliminates further analysis of the numerical stability criteria. DEPTOL affects the computer runtime and flow depth resolution. If a flood simulation yields poor results either by failing to conserve volume or by noted erratic or surging discharge, then the DEPTOL parameter may be too high for numerical stability. Decreasing DEPTOL will reduce the timestep, improve the numerical stability and longer computational times. Setting DEPTOL equal to 0 dictates that only the Courant criteria will be applied. Significant numerical instability may require revisions in slope, flow area or roughness.

3. Decreasing the maximum dynamic wave stability decreases the timestep and improves the numerical stability and volume conservation for the full dynamic wave. Review the CHANMAX.OUT file and the HYDROG program hydrograph plots for hydrograph spikes. You can also review MAXPLOT or Mapper++ or the VELTIMEFP.OUT file to determine if floodplain velocities are too high. If surging problems are observed, reduce WAVEMAX. A typical range for WAVEMAX is 0.1 to 1.0. The TIME.OUT file will identify which floodplain, channel or street elements cause the most timestep decreases as a function of the stability criteria.

4. Using the Courant criteria, the timestep Δt is limited by:

   $$\Delta t = C\,\Delta x\,/\,(\beta V + c)$$

   where:

   C is the Courant number ($C \leq 1.0$)
   Δx is the square grid element width
   V is the computed average cross section velocity
   β is a coefficient (e.g. 5/3 for a wide channel) but is seldom used
   c is the computed wave celerity

   The coefficient C may vary from 0.1 to 1.0 depending on the size of the grid element and floodwave velocity. If C is set to 1.0, artificial or numerical diffusivity is assumed to be zero. A typical value of the Courant number is 0.6 to 0.7.

Use the following approach to improve numerical stability and run-time speeds:

- Initially run the model with the Courant numbers = 0.6 and DEPTOL = 0. and WAVEMAX = 0. If the model is unstable, reduce the appropriate Courant number by 0.1 in successive runs until the Courant number reaches 0.2. If numerical instability is still observed, assigned DEPTOL and WAVEMAX values or adjust the model component attributes.
- Run the model with an appropriate limiting Froude number (e.g. FROUDL in CONT>DAT = 0.9 subcritical flow on an alluvial surface). This will calibrate the model n-values for reasonable Froude numbers.
- Review the maximum velocities in VELTIMEC.OUT, VELTIMEFP.OUT and VELTIMEST.OUT (or in MAXPLOT or Mapper++) to determine the location of any inappropriate high velocities related to numerical surging and increase the n-values in the vicinity of the grid elements with high velocities.
- Review the n-values in ROUGH.OUT and FPLAIN.RGH. Make n-value adjustments in FPLAIN.RGH based on exceedingly high n-values in ROUGH.OUT then replace FPLAIN.DAT with FPLAIN.RGH.

- Run the simulation and repeat steps 3 and 4 making adjustments to FPLAIN.RGH until ROUGH.OUT is essentially empty. A few incremental n-value changes will not affect the simulation. You can also make adjustments to FROUDL to decrease the number of n-value adjustments.

FILE: FPLAIN.DAT

FLOODPLAIN GRID ELEMENT DATA

FPLAIN.DAT File Variables

1	0	2	10	0	0.060	4005.23	Line 1:	**DUM**	**FP(I, J)**	**FP(I, 5)**	**FP(I, 6)**
2	0	3	11	1	0.065	4008.65	Line 1:	**DUM**	**FP(I, J)**	**FP(I, 5)**	**FP(I, 6)**
3	0	4	12	2	0.065	4002.23	Line 1:	**DUM**	**FP(I, J)**	**FP(I, 5)**	**FP(I, 6)**
...											
...											
...											
18	9	0	27	17	0.065	4010.78	Line 1:	**DUM**	**FP(I, J)**	**FP(I, 5)**	**FP(I, 6)**

Note: FPLAIN.DAT is a list of the grid element and its bordering grid elements. Zeros indicate boundary elements.

Example Grid

1	2	3	4	5	6	7	8	9
10	11	12	13	14	15	16	17	18
19	20	21	22	23	24	25	26	27
28	29	30	31	32	33	34	35	36

Line 1:
1 = grid element,
0 = cell to the north,
2 = cell to the east,
10 = cell to the south,
0 = cell to the west
0.060 = n-value for the cell
4005.23 = cell elevation

FPLAIN.DAT File Example

1	0	2	10	0	0.060	4005.23
2	0	3	11	1	0.065	4008.65
3	0	4	12	2	0.065	4002.23
4	0	5	13	3	0.065	4003.15
...						
33	24	34	0	32	0.065	4000.22
34	25	35	0	33	0.065	4000.56
35	26	36	0	34	0.065	4001.00
36	27	0	0	35	0.065	4001.45
...						

Variable Descriptions for the FPLAIN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DUM	i	1 - NNOD	Grid element number (I) of the floodplain grid system. This is a dummy variable that is not used by the model. It is only used for the convenience of viewing the input data file.
FP(I,J)	i	1 - NNOD	Floodplain element contiguous to grid element I (where I = 1, NNOD) and located in the J-direction (where J = 1,4). The J-direction corresponds to one of the four compass directions (see comments 1 thru 5).
FP(I,5) -	r	0.010 - 0.4	Manning's n roughness coefficient assigned to grid element I. (see comment 6)
FP(I,6)	r		Ground surface elevation for grid element I (ft or m).

Instructional Comments for the FPLAIN.DAT File

This file is prepared by the grid developer system program GDS and generally require no data revisions. Occasionally, an n-value or elevation for an individual grid element is modified. Spatial variability and editing of both the n-values and elevations are easily accomplished in the GDS programs. If grid elements are added to the FPLAIN.DAT file after creating the file, there are several rules that should be observed.

1. There should be no elements in the grid system that do not have at least one neighbor element sharing one side. In other words, no element should be connected only by a single diagonal corner.

2. The elements should be numbered consecutively starting with 1.

3. If a grid element (I) is a boundary element, then the neighboring grid element FP(I,J) where J = 1, 2, 3, or 4, is set equal to 0.

4. Any additional grid elements in the FPLAIN.DAT file must have corresponding grid elements in the CADPTS.DAT file.

5. The roughness assigned to the floodplain grid element should represent the flow resistance associated with a flow depth of 3 ft (1 m) or greater. The model automatically computes a depth variable roughness for depths less than 3 ft approximately as follows:

$$n_d = n_b * 1.5 * e^{-(0.4\ \mathrm{depth}/\mathrm{dmax})}$$

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = flow depth for drowning the roughness elements and vegetation (hardwired 3 ft or 1 m)

To turn off the depth variable roughness set AMANN = -99. See the Comment 9 in the CONT.DAT file.

FILE: MANNINGS_N.DAT

FLOODPLAIN GRID ELEMENT NVALUE DATA

MANNINGS_N.DAT File Variables

1 0.04	Line 1:	**DUM**	**FP(I, J)**
2 0.04	Line 1:	**DUM**	**FP(I, J)**
3 0.04	Line 1:	**DUM**	**FP(I, J)**
...			
...			
...			
18 0.04	Line 1:	**DUM**	**FP(I, J)**

Note: MANNINGS_N.DAT is a list of the grid elements and their n-values. This file is automatically generated by the GDS and FLO-2D model at runtime. The n-values are the same as those listed in FPLAIN.DAT when it is created or edited. Use this file for GIS or CADD applications. Combined with TOPO.DAT, it can replace the FPLAIN.DAT and CADPTS.DAT files.

MANNINGS_N.DAT File Example

```
1  0.040
2  0.040
3  0.040
4  0.040
...
33 0.040
34 0.040
35 0.040
36 0.040
...
```

Variable Descriptions for the MANNINGS_N.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DUM	i	1 - NNOD	Grid element number (I) of the floodplain grid system. This is a dummy variable that is not used by the model. It is only used for the convenience of viewing the input data file.
FPNVALUE	r	0.010 - 0.4	Manning's n roughness coefficient assigned to grid element I. (see comment 1)

Instructional Comments for the MANNINGS_N.DAT File

This file is prepared and edited by the GDS program for spatially variable n-values.

1. The elements should be numbered consecutively starting with 1.
2. The roughness assigned to the floodplain grid element should represent the flow resistance associated with a flow depth of 3 ft (1 m) or greater.
3. This file is a substitute for the n-values listed in the FPLAIN.DAT.
4. MANNING_N.DAT, MANNING_N.BAC, MANNING_N.RGH
5. This series of files is automatically generated by the Pro model and has the format of grid element number and Manning's n-value in two columns. When combined with TOPO.DAT, MANNINGS_N.DAT can be used as a substitute for FPLAIN.DAT. FPLAIN.DAT can be deleted or not used if these two files are present in the project folder. The model will recognize that either TOPO.DAT and MANNINGS_N.DAT or FPLAIN.DAT is present and will automatically generate the missing file(s). These files can be used to assigned or edit the n-values. TOPO.DAT and MANNINGS_N.DAT are in a format that is more GIS compatible and FPLAIN.DAT will eventually be obsolete. MANNINGS_N.RGH is used with the limiting Froude number component to report adjusted n-values during a simulation in place of FPLAIN.RGH.

FILE: TOPO.DAT

TOPOGRAPHICAL ELEVATION DATA

TOPO.DAT File Variables

551397.50 44608.95 6.00	Line 1:	**DUM FP(I, J)**
551397.50 44708.95 6.05	Line 1:	**DUM FP(I, J)**
551397.50 44708.95 6.06	Line 1:	**DUM FP(I, J)**
...		
...		
...		
551597.50 44808.95 7.18	Line 1:	**DUM FP(I, J)**

Note: TOPO.DAT is a list of the grid element x- and y-coordinates and their elevations. The elevations are interpolated from topographical data by the GDS. This files contains the same data as FPLAIN.DAT and CADPTS.DAT. It is automatically generated and edited by the GDS when the FPLAIN.DAT is written. Use this file for compatible GIS and CADD applications.

TOPO.DAT File Example

```
551397.50 44608.95 6.00
551397.50 44708.95 6.05
551397.50 44808.95 6.06
551397.50 44908.95 6.06
551397.50 45008.95 6.11
551397.50 45108.95 6.09
551397.50 45208.95 6.12
551397.50 45308.95 6.14
...
...
```

Variable Descriptions for the TOPO.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
XCOORD(I)	r	∞	X coordinate of grid element node at center.
YCOORD(I)	r	∞	Y coordinate of grid element node at center.
ELEV(I)	r	∞	Elevation of grid element.

Instructional Comments for the TOPO.DAT File

1. The data in this file is the same as that in FPLAIN.DAT and CADPTS.DAT and is in a format that enables GIS and CADD applications to use it directly.

2. The TOPO.DAT and MANNINGS_N.DAT files replace FPLAIN.DAT and CADPTS.DAT files. If these files are generated by GIS and CADD programs, the FLO-2D model can run without the FPLAIN.DAT and CADPTS.DAT if the data is space delimited.

3. TOPO.DAT and TOPO.BAC.

4. Combined with MANNINGS_N.DAT, TOPO.DAT replaces FPLAIN.DAT. TOPO.DAT has the format of x- and y-coordinate, and elevation (x,y,z file) of the grid element center with the grid elements list from 1 to the end. It is more GIS compatible than FPLAIN.DAT. If TOPO.DAT is missing at runtime, the model automatically generates this file. Similarly if FPLAIN.DAT is missing at runtime, the model automatically generates this file. Eventually FPLAIN. DAT will be obsolete.

FILE: INFLOW.DAT

INFLOW HYDROGRAPH DATA

INFLOW.DAT File Variables

0 4335	Line 1:	**IHOURDAILY IDEPLT**
C 0 4335	Line 2:	**IFC(I) = 'F' or 'C' INOUTFC(I) KHIN(I)**
		I = Number of inflow nodes.
H 0 0	Line 3:	**HYDCHAR = 'H' HP(J,1) HP(J,2) HP(J,3) J=1**
H 1 5530	Line 3:	**HYDCHAR = 'H' HP(J,1) HP(J,2) HP(J,3) J=2**
H 24 5530	Line 3:	**HYDCHAR = 'H' HP(J,1) HP(J,2) HP(J,3) J=3**
...		
R 5232 3320	Line 4:	**RESCHAR = 'R' IRESGRID(J) RESERVOIREL(J) J=1**
R 6528 3295	Line 4:	**RESCHAR = 'R' IRESGRID(J) RESERVOIREL(J) J=2**
		J = Number of data pairs.
....		

Notes:
If only rainfall is being simulated omit this file
Line 2, 3: Repeat these lines for each inflow grid element.
Line 3: If MUD = 0, HP(I,J,3) is omitted.

INFLOW.DAT File Example

```
0  4335
C        0   4335
H        0    0
H        1   5530
H        2   5530
H        3   5378
H        4   5378
H        5   5227
H        6   5075
H        7   4924
R     5232
   ....
```

Variable Descriptions for the INFLOW.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
HP(I,J,1)	r	0.0 - ∞	Time corresponding to the start of the floodplain inflow hydrograph interval (hours or days). The first hydrograph time-discharge set should be 0.0 and 0.0.
HP(I,J,2)	r	0.0 - ∞	Floodplain discharge (cfs or m^3) corresponding to the time interval which starts at HP(I,J,1).
HP(I,J,3)	r	0 - 1	Sediment concentration by volume (see comment 2).
HYDCHAR	c	H	Character 'H' that identifies Line 3 inflow hydrograph time and discharge pairs. Each line of the hydrograph begins with 'H'.
IDEPLT	i	1 - NNOD	Inflow grid element number whose hydrograph is to be graphically displayed at runtime. Only one inflow grid element hydrograph can be plotted on the screen. If no graphic display is desired (LGPLOT = 0) set IDEPLT = 0 (see comment 3).
IFC(I)	c	F or C	Character 'F' or 'C' to identify the inflow hydrograph grid element as a floodplain 'F' or a channel 'C (see comment 1).
INOUTFC(I)	s	0 = inflow 1 = outflow	INOUT = 0 for inflow hydrograph. INOUT = 1 for outflow hydrograph or diversion from the channel (see comment 4).
IHOURDAILY	s	0 = hourly 1 = daily	IHOURDAILY = 0 for hourly intervals in the discretized inflow hydrograph HP (I,J,1). IHOURDAILY = 1 for (24hr) intervals of HP (I,J,1).
KHIN(I)	i	1 - NNOD	Array of grid elements with a inflow hydrograph (inflow nodes).
RESCHAR	c	R	Character 'R' that identifies Line 4 for reservoir or ponded area water surface assignment.
IRESGRID	i	1 - NNOD	Grid element located somewhere inside the reservoir or ponded water area. Only one grid element has to be assigned a water surface elevation (see comment 5).
RESERVOIREL	r	0 - 30,000 0 - 9,000	Water surface elevation (ft or m) of the reservoir or ponded water area.

Instructional Comments for the INFLOW.DAT File

1. Either the channel or the floodplain grid elements can be used to input the inflow hydrograph to grid system.

2. The user has a choice to input either the sediment concentration by volume associated with the inflow water discharge or a sediment volume for the time interval starting at HP(I,J,1). The sediment volume (ft^3 or m^3) can represent erosion, hillslope failure, or any other type of mass sediment loading. When HP(I,J,3) is less than 1.0, HP (I,J,3) corresponds to the sediment concentration by volume for floodplain discharge HP(I,J,2) for the time interval which starts at HP(I,J,1). If HP(I,J,3) is greater than 1.0, then HP(I,J,3) represents a sediment inflow volume

3. IDEPLT must be an inflow grid element KHIN(I) listed in Line 2.

4. If the channel inflow hydrograph is to be plotted at runtime on the screen, set LGPLOT = 2 in the CONT.DAT file.

5. To create a filled reservoir or pond, simply assign the desired water surface elevation to one grid element (IRESGRID) within the ponded area. At runtime, the model will automatically assign the same water surface to all the grid elements that are contiguous to IRESGRID that a have a ground elevation less than the prescribed water surface elevation RESERVOIREL.

FILE: OUTFLOW.DAT

OUTFLOW HYDROGRAPH DATA

OUTFLOW.DAT File Variables

K 374	Line 1:	**OUTCHAR = 'K' KOUT**
H 10.0 2.6 0.35	Line 2:	**OUTCHAR = 'H' HOUT(J,1) HOUT(J,2) HOUT(J,3)**
K 1007	Line 1:	**OUTCHAR = 'K' KOUT**
T 0.0 0.00	Line 3:	**OUTCHAR = 'T' CHDEPTH(J) CQTABLE(J) J=1**
T 3.0 50.35	Line 3:	**OUTCHAR = 'T' CHDEPTH(J) CQTABLE(J) J=2**
T 5.0 157.67	Line 3:	**OUTCHAR = 'T' CHDEPTH(J) CQTABLE(J) J=3**
K 567	Line 1:	**OUTCHAR = 'K' KOUT**
N 567 1	Line 4:	**OUTCHAR = 'N' NOSTA NOSTACFP**
S 0.00 0.00	Line 5:	**OUTCHAR = 'S' STA_TIME(J) STA_STAGE(J) J=1**
S 0.50 10.00	Line 5:	**OUTCHAR = 'S' STA_TIME(J) STA_STAGE(J) J=2**
S 1.00 20.00	Line 5:	**OUTCHAR = 'S' STA_TIME(J) STA_STAGE(J) J=3**
O 273	Line 6	**OUTCHAR = 'O' NODDC(J) J=1**
O1 373	Line 6:	**OUTCHAR = 'O1' NODDC(J) J=2**
O2 374	Line 6:	**OUTCHAR = 'O2' NODDC(J) J=3**
O3 567	Line 6:	**OUTCHAR = 'O3' NODDC(J) J=4**
		J = number of parameters

Notes:

Line 1, 2 and 3: If channel = 0 in CONT.DAT omit these lines.

Line 1: Repeat for each channel outflow element.

Line 2: Omit line if no stage-discharge control relationship is required for the channel outflow.

Line 3: Omit line if no stage-discharge control is required for the channel outflow. If Lines 2 and 3 are omitted, the channel outflow will be discharge from the grid system as normal flow.

Line 4 and 5: Repeat lines for each element with a time-stage relationship.

Line 6: Repeat for each floodplain outflow grid element and each outflow node that will generate a hydrograph.

FILE: OUTFLOW.DAT

OUTFLOW HYDROGRAPH DATA

OUTFLOW.DAT File Example

```
K    374
H   10.0   2.6   0.35
K   1007
T    0.0       0.00
T    3.0      50.35
T    5.0     157.67
T   10.0     366.58
K    567
N        567   1
S      0.00   0.00
S      0.50   10.00
S      1.00   20.00
S      1.50   10.00
S      2.00    0.00
O    273
O    373
O    374
O    564
O    565
O    566
O    566
O    567
O    568
O1   1005
O1   1006
O1   1007.....
```

Variable Descriptions for the OUTFLOW.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
CTDEPTH(J)	r	0.0 - ∞	Array of channel maximum depths above the thalweg (not water surface elevation) for the outflow rating table.
CQTABLE(J)	r	0.0 - ∞	Array of discharges for the channel outflow rating table.
HOUT(J,1)	r	0.01 - ∞	Array of channel maximum depths for which a channel outflow stage-discharge relationship is valid.
HOUT(J,2)	r	0.0 - ∞	Array of coefficients for the channel element outflow stage-discharge relationship (see comment 3).
HOUT(J,3)	r	0.0 - ∞	Array of exponents for the channel element (I) outflow stage-discharge relationships
KOUT	i	1 - NNOD	Array of channel outflow elements. These elements discharge flow out of the grid system from the channel (see comments 1 and 2)
NODDC	i	1 - NDC	Array of floodplain outflow grid elements. These elements discharge flow out of the grid system from the floodplain (see comments 1 and 2).
NOSTA	i	1 - NNOD	Array of grid elements with stage-time relationships. If NOSTA is a inflow element, assign NOSTA as a negative value to compute inflow volume (see comments 4, 5 and 6).
NOSTACFP	s	0 = flood-plain 1 = channel	Channel or floodplain identifier. If NOSTACFP = 0, the following stage-time relationship is for a floodplain element. If NOSTACFP = 1, the stage-time relationship is for a channel element.
OUTCHAR	c	K, H, T, N, S, O O1 - O9	Character line identifier that initializes each line in the data file. See Comment 7.
STA_TIME(J)	r	0.0 - ∞ 500 pairs	Array of time intervals (hrs) for the grid element stage-time relationship.
STA_STAGE(J)	r	0.0 - ∞ 500 pairs	Array of water surface elevations (ft or m) for the stage-time relationship.

Instructional Comments for the OUTFLOW.DAT File

1. Either the channel or the floodplain outflow elements can be used to discharge the flow from the grid system. The outflow node is an artificial grid element whose sole purpose is to discharge flow off the grid system. The outflow nodes should not contain hydraulic structures, streets or other attributes. The floodplain elevation of the outflow node is automatically set to an elevation lower (0.25 ft or 0.1 m) than the lowest upstream grid element unless it is already lower than all the upstream grid elements.

2. Omitting Lines 2 and 3 will cause all the inflow to the outflow elements to discharge from the grid system at normal flow conditions. This outflow is equal to the sum of the inflow from the contiguous elements that are not outflow nodes. This method enables an approximation of normal flow depth in the outflow elements. This is a simple method to ensure that backwater related to arbitrary boundary conditions does not occur in the upstream elements.

3. Channel boundary outflow condition may be established by specifying a stage-discharge relationship given by $Q = a\ h^b$ where the coefficient (a) and exponent (b) are required input and h is the flow depth. The coefficient (a) and exponent (b) can be used to established critical flow at the outflow grids.

4. A discretized time-stage relationship can be employed to specify a water surface elevation for at various channel or floodplain locations in the grid system. This is a simple method by which to simulate storm surge flooding on the coastal floodplain. Floodplain or channel elements can be specified with increasing tides or surge water surface elevations.

5. If a coastal flooding (storm surges or tsunamis) is being simulated with a time-stage hydraulic control, you can assign the time-stage control to the outflow nodes. When the time-stage water surface elevation in OUTFLOW.DAT is higher than the model predicted stage, inflow to the grid system will occur with assigned time-stage elevation to the outflow node. If the model predicted water surface is higher than the assigned time-stage elevation, the grid element will function as an outflow node discharging flow off the grid system. It is permissible to assign NOSTA time-stage control to grid elements that are not outflow nodes.

6. If a water surface elevation is specified for a NOSTA element, determine if it is an inflow element in the INFLOW.DAT file. If NOSTA is an inflow element, set NOSTA as negative value to compute the inflow volume at this element which corresponds to the constant water surface elevation.

7. If the OUTCHAR is O1-O9, these outflow grid elements will generate hydrographs that can be used as inflow hydrographs to a separate downstream FLO-2D model with a different grid system (even if the downstream system has a different element size). The inflow hydrograph will be in the format of the INFLOW.DAT file. This enables a row or column of outflow grid elements to be defined as inflow elements to the downstream grid system. Up to nine separate additional grid systems can be used. If only one downstream grid system will have the inflow hydrographs, set OUTCHAR = O1 for those boundary outflow nodes. The CADPTS.DAT file for the downstream grid system must be included in the project folder as CADPTSDS1.

FILE: RAIN.DAT

RAINFALL DATA

RAIN.DAT File Variables

0 0	Line 1:	**IRAINREAL, IRAINBUILDING**
3.100 0.000 0 1	Line 2:	**RTT RAINABS RAINARF MOVINGSTORM**
R 0.000 0.000	Line 3:	**RAINCHAR = 'R' R_TIME(I) R_DISTR(I) I=1**
R 0.083 0.050	Line 3:	**RAINCHAR = 'R' R_TIME(I) R_DISTR(I) I=2**
R 0.167 0.110	Line 3:	**RAINCHAR = 'R' R_TIME(I) R_DISTR(I) I=3**
R 0.250 0.300	Line 3:	**RAINCHAR = 'R' R_TIME(I) R_DISTR(I) I=4**
R 0.330 0.450	Line 3:	**RAINCHAR = 'R' R_TIME(I) R_DISTR(I) I=5**
R....		
2.0 5	Line 4:	**RAINSPEED IRAINDIR**
2558 0.5	Line 5	**IRGRID(I) RAINARF(I)**
		I = number of rainfall depth-area reduction values

Notes:

Line 4: If MOVINGSTORM = 0, omit this line.

Line 5: If IRAINARF = 0, omit this line

RAIN.DAT File Example

```
0   0
3.100    0.000    1    1
R  0.000      0.000
R  0.083      0.050
R  0.167      0.110
R  0.250      0.300
R  0.330      0.450
R....
2.0  5
2558  0.50
```

Variable Descriptions for the RAIN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
IRAINARF	s	0 = off 1 = on	IRAINARF = 1 indicates that individual grid element depth-area reduction values will be assigned.
IRAINBUILDING	s	0 = off 1 = on	IRAINBUILDING = 1 indicates that rainfall on the ARF portion of the grid element will be contributed to the surface water runoff for that element. See comment 3.
IRAINDIR	i	1 thru 8	Direction of the moving storm. Directions are as follows: 1 = N 5 = NE 2 = E 6 = SE 3 = S 7 = SW 4 = W 8 = NW
IRAINREAL	s	0 = off 1 = on	IRAINREAL = 1 indicates that real-time rainfall (e.g. NEXRAD) will be simulated. The RAINCELL.DAT file containing the spatial and temporal rainfall data must be prepared by the GDS.
IRGRID	i	1 - NNOD	Grid element with a spatially defined rainfall depth area reduction value. This data is automatically generated in the GDS processor program
MOVINGSTORM	s	0 = off 1 = on	MOVINGSTORM = 1 indicates that a moving storm will be simulated.
RAINABS	r	0 - 1	Rainfall interception and abstraction (inches or mm) if infiltration is not being modeled (see comment 2).
RAINARF	r	0 - 1	Rainfall depth area reduction to create spatially variable rainfall. This data is automatically generated in the GDS processor program.
RAINCHAR	c	R	Character 'R' that identifies Line 3.
RAINSPEED	r	0 - 100 0 - 50	Storm speed (mph or mps)
RTT	r	0.0 - ∞	Total storm rainfall (inches or mm).
R_TIME(I)		0.0 - ∞	Time (hours) corresponding to the start of the specified rainfall interval.
R_DISTR(I)	r	0 - 1	Rainfall distribution as a cumulative percentage of the total storm which initiates at the time interval R_TIME(I) (see comment 1)..

Instructional Comments for the RAIN.DAT File

1. Rainfall distribution varies by 0.0 < R_DISTR(I) < 1.0. The rainfall distribution has to be related to the overall flood simulation time. The rainfall may occur for only a portion of the total flood simulation and may start later than the flood simulation time represents. For most rain storms, the start of the simulation correlates with the start of the rainfall. In those cases where the rainfall and the simulation time are not correlated, it may be necessary to use 0.0 cumulative rainfall at the beginning of the flood simulation for a period of time and/or similarly 1.0 (100%) cumulative rainfall near the end of the simulation.

2. If infiltration is being simulated, set the RAINABS = 0 and assign the rainfall abstraction in the INFIL.DAT file.

3. When rainfall occurs on a grid element with a storage loss assigned (ARF value), the default condition is that them model removes that volume from the surface water in that cell. It assumes that the rainfall on buildings enters the storm drain system and is eliminated as runoff. Setting IRAINBUILDING = 1 enables the model to add the rainfall to the surface water of the grid element with an ARF value. It assumes that the buildings have a gutter system that discharges the water to the ground.

0 = no infiltration

looks for data - spatially

does default 10' + 3m

>.01

or

<.01 > 0

FILE: INFIL.DAT

INFILTRATION DATA

INFIL.DAT File Variables

3	Line 1:	**INFMETHOD**
0 0.7 1 0.4 10.0 1	Line 2:	**ABSTR SATI SATF POROS SOILD INFCHAN**
0.1 4.3 0	Line 3:	**HYDCALL SOILALL HYDCADJ**
0.03	Line 4:	**HYDCXX *See Notes**
R 0.03	Line 4a:	**INFILCHAR = 'R' HYDCX(IC) *See Notes**
R 0.03 0.3 10.0	Line 4b:	**INFILCHAR = 'R' HYDCX(IC) HYDCXFINAL(IC) SOIL_DEPTHCX(IC)** *IC= number of channel segments or reaches*
99 0	Line 5:	**SCSNALL ABSTRSCS**
F 1730 0.01 4.3 0.3 0.0 0.0 10.0	Line 6:	**INFILCHAR = 'F' INFGRID(N) HYDC(N) SOILS(N) DTHETA(N) ABSTRINF(N) RTIMPF(N) SOIL_DEPTH(N)** *N = 1 - number of infiltration data sets*
S 320 82.00	Line 7:	**INFILCHAR = 'S' INFGRID(N) SCSCN(N)**
C 2 0.04	Line 8:	**INFILCHAR = 'C' INFCH(N) HYDCONCH(N)**
I 5.0 1.0 0.0007	Line 9:	**INFILCHAR = 'I' FHORTONI FHORTONF DECAYA**
H 3.0 0.5 0.00018	Line 10:	**INFILCHAR = 'H' INFGRID(IF) FHORTI(INFGRID(IF)) FHORTF(INFGRID(IF)) DECA(INFGRID(IF))** *IF = 1 - number of Horton infiltration elements*

Notes:

If INFIL = 0 in the CONT.DAT file, omit this file.
If INFMETHOD = 1 (Green-Ampt) add Line 2 thru 4, skip Line 5. Line 6 is optional.
If INFMETHOD = 2 (SCS Curve) add Line 5, skip Lines 2 thru 4. Line 7 is optional.
If INFMETHOD = 3 (Both Green-Ampt and SCS) add Lines 2 thru 5. Line 6 and 7 are optional.
If INFMETHOD = 4 (Horton), add lines 9 and 10.

*If INFCHAN = 1 add Line 4. Line 8 is optional.
If SOILD = 0. Use Line 4 or 4a
If SOILD > 0. use Line 4b
Line 4a or 4b, use one line per reach.

FILE: INFIL.DAT - CONT.

INFILTRATION DATA

INFIL.DAT File Example

```
3
0  0.7  1  0.4  10.0  1
0.1  4.3
R  0.03  0.2  5.0
R  0.12  0.3  10.0
99  0
F  1730  1.01  4.3  0.3  0.0  0.0  8.5
F  1730  1.01  4.3  0.3  0.0  0.0  10.0
F  1730  1.01  4.3  0.3  0.0  0.0  9.2
F...
S  320  82.00
S  321  82.00
S  322  82.50
S...
C  2  0.04
C  10  0.04
C...
```

Variable Descriptions for the INFIL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ABSTR	r	0 - 1 0 - 25	Green Ampt global floodplain rainfall abstraction or interception (inches or mm) (see comments 1 and 7).
ABSTRINF(N)	r	0 - 1 0 - 25	Individual floodplain grid element rainfall abstraction (inches or mm)
ABSTSCS	r	0 - 1 0 - 25	SCS global floodplain rainfall abstraction or interception (inches or mm). Assign ABSTRSCS = 0 for automatic computation of the initial abstraction (see comments 7 and 10).
DECA (INFGRID(IF))	r	0.0007 - 0.00018	Horton's equation spatially variable decay coefficient (1/second; no metric). (see comment 17)
DECAYA	r	0.0007 - 0.00018	Horton's equation decay coefficient (1/second; no metric). (see comment 17)
DTHETA(N)	r	0.0 - 1 0.0 - 0.5	The individual floodplain grid element soil moisture deficit (SATF-SATI) is expressed as a decimal with a range from 0.0 to 1.0. It can also represent the individual floodplain grid element volumetric soil moisture deficit that is defined as the soil moisture deficit multiplied by the porosity (SATF-SATI)*POROS with a practical range from 0.0 to 0.5 (see comment 11). Set POROS = 0 for the volumetric soil moisture deficiency.
DTHETAC(I)	r	0.0 - 1 0.0 - 0.5	The channel segment or reach soil moisture deficit (SATF-SATI) is expressed as a decimal with a range from 0.0 to 1.0. It can also represent the channel reach volumetric soil moisture deficit that is defined as the soil moisture deficit multiplied by the porosity (SATF-SATI)*POROS with a practical range from 0.0 to 0.5 (see comment 11). Set POROS = 0 for the volumetric soil moisture deficiency.
FHORTF (INFGRID(IF))	r	0.5 - 1.0	Horton's equation spatially variable floodplain final infiltration rate (inches/hr; no metric).
FHORTI (INFGRID(IF))	r	3.0 - 5.0	Horton's equation spatially variable floodplain initial infiltration rate (inches/hr, no metric).
FHORTONF	r	0.5 - 1.0	Global Horton's equation final infiltration rate (inches/hr; no metric)

Variable Descriptions for the INFIL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
FHORTONI	r	3.0 - 5.0	Global Horton's equation initial infiltration rate (inches/hr; no metric) (see comment 13)
HYDC(N)	r	0.01 - 10 0.25 - 250	Average hydraulic conductivity of an individual floodplain grid element (inches/hr or mm/hr) (see comments 2, 4 and 5).
HYDCALL	r	0.01 - 10 0.25 - 250	Average global floodplain hydraulic conductivity (inches/hr or mm/hr).
HYDCADJ	r	0.0 - ∞	Hydraulic conductivity adjustment variable for spatially variable hydraulic conductivity. If HYDCADJ < 1.00 HYDCS(i) = HYDCS(i) + HYDCADJ If HYDCADJ > 1.00 HYDCS(i) = HYDCS(i) * HYDCADJ
HYDCHN	r	0.01 - 10 0.25 - 250	Average global hydraulic conductivity for the entire channel (inches/hr or mm/hr) (see comment 8).
HYDCHN(I)	r	0.01 - 10 0.25 - 250	Channel reach hydraulic conductivity channel (inches/hr or mm/hr) (see comment 8).
HYDCONCH(N)	r	0.01 - 10 0.25 - 250	Hydraulic conductivity for a channel element (inches/hr or mm/hr).
HYDCX(IC)	r	0.01 - 10 0.25 - 250	Initial hydraulic conductivity for a channel segment (inches/hr or mm/hr) (see comment 18).
HYDCXFINAL(IC)	r	0.01 - 10 0.25 - 250	Final hydraulic conductivity for a channel segment (inches/hr or mm/hr).
INFCH(N)	i	1 - NNOD	Array of channel elements with a unique hydraulic conductivity
INFCHAN	s	0 = off 1 = on	Set switch to 1 to simulate channel infiltration (see comment 6).
INFGRID(IF)	i	1 - NNOD	Array of floodplain grid elements with individual infiltration parameters (see comment 3).

Variable Descriptions for the INFIL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
INFILCHAR(N)	c	F, S, C, R, I, H	'F' for spatially variable floodplain Green-Ampt data (Line 6), 'S' for floodplain spatially variable SCS curve number data (Line 7); 'C' for channel spatially variable channel infiltration data, (Line 8). 'R' for channel reach infiltration data (Line 4 and 4a); 'I' for Horton global parameters (Line 9); 'H' for Horton spatially variable floodplain parameters (Line 10).
INFMETHOD	s	1, 2, 3 or 4	Set to 1 to simulate infiltration with the Green-Ampt method; Set to 2 to simulate infiltration with the SCS curve number method; Set to 3 for both; Set to 4 for Horton infiltration.
POROS	r	0.3 - 0.5	Global floodplain soil porosity. If using the volumetric soil moisture deficiency for DTHETA, set POROS = 0.
RTIMPF(N)	r	0.0 - 1	Percent impervious floodplain area on a grid element.
SATF	r	0.5 - 1	Global final saturation of the soil (in percent) for computing infiltration.
SATI	r	0.0 - 0.95	Global initial saturation of the soil (in percent).
SCSNALL	r	1. - 100.	Global floodplain SCS curve number for infiltration (see comment 9).
SCSN(N)	r	1. - 100.	SCS curve numbers for spatially variable infiltration of the floodplain grid elements.
SOIL_DEPTH(N)	r	0.0 - 100.	Spatially variable Green-Ampt infiltration soil limiting depth storage (ft or m). Maximum soil depth for infiltration on a grid element. See comment 12.
SOIL_DEPTHCX(IC)	r	0.0 - 100.	Maximum soil depth for the initial channel infiltration. When SOIL_DEPTHCX is exceeded, the exponential decay from the initial hydraulic conductivity to the final hydraulic conductivity begins. See comment 18.

Variable Descriptions for the INFIL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
SOILD	r	0.0 - 100.	Global Green-Ampt infiltration soil limiting depth storage (ft or m). Maximum soil depth for infiltration. Set SOILD = 0 to have unlimited infiltration and do not assign spatially variable SOIL_DEPTH(N).
SOILS(N)	r	1 - 20 25 - 500	Capillary suction head for floodplain grid elements (inches or mm).
SOILALL	r	1 - 20 25 - 500	Average global floodplain capillary suction head (inches or mm).

Instructional Comments for the INFIL.DAT File

1. The Green-Ampt infiltration parameters including hydraulic conductivity HYDC, initial abstraction ABSTR, initial saturation SATI, and soil capillary suction head SOILS, can be estimated from the tables in the FLO-2D Reference Manual (Tables 3-6). Generally, the final SATF can be set at 100% and the porosity can be assumed to be 0.4.

2. No infiltration is simulated if the sediment concentration by volume is greater than 10%. This precludes simulating infiltration during mudflows.

3. The individual floodplain grid elements with unique Green-Ampt infiltration parameters are specified in Line 6. Line 6 infiltration parameters supercede the values in Line 2.

4. No infiltration is computed for the portion of the grid element removed from the potential flow surface with an Area Reduction Factor (ARF). No infiltration is computed for grid elements that are completely removed from the potential flow surface (ARF = 1.0). Rainfall, however, is assumed to occur over the entire grid element even if the element is completely blocked out by setting ARF = 1. Increased runoff resulting from proposed development can be predicted by using the ARF values to limit infiltration on a grid element.

5. No infiltration is computed for street areas of a grid element. The street area is subtracted from the overland portion of the grid system.

6. Channel infiltration is computed only if INFCHAN = 1. Generally channel infiltration is negligible for channels with perennial flow. The simulation of channel infiltration may be important for small flood events in ephemeral alluvial fan channels with porous bed material.

7. Precipitation abstraction is an initial loss of rainfall that precedes infiltration and excess rainfall runoff. Vegetation interception is a component of the initial loss. Abstraction values will generally range from 0.01 to 0.5 inches. In addition, FLO-2D does not initiate any flood routing until the depression storage TOL is filled. The TOL value is specified in TOLER.DAT file. Abstraction is often assumed to include depression storage, but in FLO-2D a value of 0.1 ft is generally assumed for TOL. This TOL depression storage should be considered when estimating the abstraction.

8. Use HYDCX(IC) and all other parameters on Line 4 to specify channel infiltration data by reach. Use line 8 HYDCON parameter to specify spatially variable hydraulic conductivity in the channel grid elements that will supersede the

HYDCX(IC) value in Line 4. It is not necessary to specify individual channel element soil suction, initial or final saturation values when assigning channel infiltration. If SOILD is = 0, use Line 4, where IC is the number of channel segments or reaches each entered on a new line. If SOILD is greater than 0, use line 4a where IC is the number of segments or reaches.

9. If you simulate infiltration with the SCS curve number method (INFMETHOD = 2), it is assumed that the channel infiltration is negligible. You can only simulate channel infiltration with the Green-Ampt method.

10. If you simulate infiltration with the SCS curve number method (INFMETHOD = 2), assign the ABSTRSCS variable in Line 5 to the abstraction (inches or mm). If ABSTRSCS = 0.0, the abstraction value is automatically computed using the SCS method.

11. The infiltration parameters can be estimated from the tables in the Reference Manual. The user must distinguish whether soil moisture deficit parameter DTHETA will represent the volumetric soil moisture deficit (soil moisture deficit times the porosity) as prescribed from a drainage manual or if DTHETA will be defined as just the soil moisture deficit (SATF-SATI). If the volumetric soil moisture deficit (SATF-SATI)*POROS is being applied, set POROS = 0.0 and assign a DTHETA value in the range from 0.0 to 0.5. If the only soil moisture deficit is being used, then assign a typical porosity (POROS) in the range: 0.35 to 0.45.

12. The Green-Ampt infiltration will cease when the wetting front reaches the limiting soil depth either SOILD, SOIL_DEPTH or SOIL_DEPTHCX for the channel.

13. It is not necessary to specify the soil suction, initial or final saturation values when simulating channel infiltration. These values are assumed not to be important to the channel bed or bank infiltration.

14. If you simulate infiltration with the SCS curve number method (INFMETHOD = 2), it is assumed that the channel infiltration is negligible. You can only simulate channel infiltration with the Green-Ampt method.

15. If you simulate infiltration with the SCS curve number method (INFMETHOD = 2), assign the ABSTRSCS variable in Line 5 to the abstraction (inches or mm). If ABSTRSCS = 0.0, the abstraction value is automatically computed using the SCS method.

16. The infiltration parameters can be estimated from the tables in the Reference Manual. The user must distinguish whether soil moisture deficit parameter DTHETA will represent the volumetric soil moisture deficit (soil moisture deficit times the porosity) as prescribed from a drainage manual or if DTHETA will be defined as just the soil moisture deficit (SATF-SATI). If the volumetric

soil moisture deficit (SATF-SATI)*POROS is being applied, set POROS = 0.0 and assign a DTHETA value in the range from 0.0 to 0.5. If the only soil moisture deficit is being used, then assign a typical porosity (POROS) in the range: 0.35 to 0.45.

17. Horton's infiltration model is defined by the equation:

$$f = f_n + (f_i - f_n)\, e^{-at}$$

Where:

f = infiltration rate at simulation time t from start of the rainfall

f_i = initial infiltration rate (in/hr)

f_n = final infiltration rate (in/hr)

a = decay coefficient (1/sec)

t = time from start of rainfall (sec)

There are no metric equivalent values. This method is based on the NRCS hydrologic groups in English Units.

18. As the channel infiltration storage fills, the infiltration rate declines but does not cease. The decay of the hydraulic conductivity H_c from the initially assigned hydraulic conductivity H_i to a final saturated hydraulic conductivity H_f is based on the following equation:

$$H_c = H^f + (H_i - H_f)\, e^{-at}$$

where:

a = decay coefficient hardwired to 0.00002, selected to have the decay from the initial to the final hydraulic conductivity over a 72 hr period with the decay to half the original hydraulic conductivity in 12 hours.

t = time (seconds) from when the wetting front reaches the limiting soild depth

FILE: EVAPOR.DAT
EVAPORATION DATA

EVAPOR.DAT File Variables

5 1 0.00	Line 1: **IEVAPMONTH IDAY CLOCKTIME**
january 2	Line 2: **EMONTH(I) EVAP(I)** *I = 1 - 12*
0.0071	Line 3: **EVAPER(I, J)** *I = 1, 12 Months, J = 1,24 Hours*
0.0086	Line 3: **EVAPER(I, J)** *I = 1, 12 Months, J = 1,24 Hours*
0.0051	Line 3: **EVAPER(I, J)** *I = 1, 12 Months, J = 1,24 Hours*
...	

Notes:
If IEVAP = 0 in the CONT.DAT file, omit this file.
Line 3: Repeat 24 times for every Line 2.

An example of the EVAPOR.DAT file is available in the FLO-2D Example Project subdirectory based on available data for the Rio Grande.

EVAPOR.DAT File Example

```
5  1  0.00
 january  2.00
   0.0071
   0.0086
   0.0051
   0.0065
   0.0038
   0.0040
   0.0055
   0.0090
   0.0285
   0.0556
   0.0799
   0.0975
   0.1154
   ...
```

FILE: CHAN.DAT

CHANNEL DATA

CHAN.DAT File Example

```
  0.00   0.60   0.40
R  3170  4433.00 4433.00 0.032 40.00  9.30 520.00
R  3118  4431.00 4431.00 0.032 20.00  9.50 510.00
R  3066  4430.30 4430.30 0.032 35.00 11.00 500.00
R  3013  4430.00 4430.00 0.032 35.00 12.70 500.00
R  ...
  0.00   0.70   0.20
V  4560 4675.19 4675.19 0.060 10.59 550.00 36.774  1.630 63.369 0.491 63.261  0.486  0.00
V  4385 4673.10 4673.10 0.050 11.00 620.00 30.774  1.630 63.369 0.491 63.261  0.486  0.00
V  4212 4672.86 4672.86 0.040 13.56 560.00 24.439  1.905 53.016 0.749 42.886  0.745  0.00
V  4213 4672.46 4672.46 0.040 16.16 550.00 22.200  1.807 31.248 0.696 31.235  0.688  0.00
V  ...
  0.00   0.60   0.40
T  7170  4423.00 4423.00 0.032 40.00  9.30 520.00 1.60 1.90
T  7118  4421.00 4421.00 0.032 20.00  9.50 510.00 2.60 2.70
T  7066  4420.30 4420.30 0.032 35.00 11.00 500.00 1.60 1.20
T  7013  4420.00 4420.00 0.032 35.00 12.70 500.00 1.60 1.20
T  ...
 -1.00   0.60   0.20   5
N  7432 0.060  450.00    1
N  7389 0.059  450.00    2
N  7344 0.050  590.00    3
N  7298 0.060  590.00    4
N  7299 0.060  590.00    5
N  ...
7432  4432.00
7160  4427.00
C  3669  3825
C  6296  6377
C  ...
E  7346
E  7345
E  7344
E  7343
E  ...
```

Variable Descriptions for the CHAN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
A1(I)	r	0.0 - ∞	Array of coefficients for the variable area regression relationships (see comment 5).
A2(I)	r	0.0 - ∞	Array of exponents for the variable area regression relationships (see comment 5).
A11(I)	r	0.0 - ∞	Array of coefficients for the variable area regression relationships for flow depth above EXCDEP(I) (see comment 5).
A22(I)	r	0.0 - ∞	Array of exponents for the variable area regression relationships for flow depth above EXCDEP(I) (see comment 5).
B1(I)	r	0.0 - ∞	Array of coefficients for the variable wetted perimeter relationships (see comment 5).
B2(I)	r	0.0 - ∞	Array of exponents for the variable wetted perimeter relationships (see comment 5).
B11(I)	r	0.0 - ∞	Array of coefficients for the variable wetted perimeter relationships for flow above EXCDEP(I) (see comment 5).
B22(I)	r	0.0 - ∞	Array of exponents for the variable wetted perimeter relationships for flow above EXCDEP(I) (see comment 5).
BANKELR(I)	r	0.01 - ∞	Right bank elevation looking downstream(see comment 12).
BANKELL(I)	r	0.01 - ∞	Left bank elevation looking downstream (see comment 12).
C1(I)	r	0.0 - ∞	Array of coefficients for the variable top width relationships (see comment 5).
C2(I)	r	0.0 - ∞	Array of exponents for the variable top width relationships (see comment 5).
C11(I)	r	0.0 - ∞	Array of coefficients for the variable top width relationships for flow depth above EXCDEP(I) (see comment 5).
C22(I)	r	0.0 - ∞	Array of exponents for the variable top width relationships for flow depth above EXCDEP(I) (see comment 5).
CHANCHAR	c	F, E	Character line identifier for ICONFLO 'C' and NOEXCHANGE 'E' channel elements.
ICONFLO1(J)	i	1 - NNOD	Tributary channel element at the confluence. See comment 8.
ICONFLO2(J)	i	1 - NNOD	Main channel element at the confluence.

Variable Descriptions for the CHAN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DEPINITIAL(K)	r	0.0 - ∞ or -1	Set DEPINITIAL(K) to the initial flow depth for the all channel elements in the given channel segment. Set DEPINITIAL = 0. for no initial channel flow depth. If DEPiNITIAL is to be used to specify an initial water surface, set DEPINITIAL = -1 and include Line 3 (see comment 2).
EXCDEP(I)	r		Array of channel depths above which a second variable area relationship will apply (see comment 4). If only one channel geometry relationship is used, set EXCDEP(I) = 0.
FCN(I)	r	0.01 - 0.25	Average Manning's n roughness coefficient for the portion of the channel in the grid element ICHANGRID (see comment 6).
FCD(I)	r	.01 - 1000	Thalweg channel depth (ft or m) for the portion of channel within the grid element. The thalweg depth is the deepest part of the channel measured from the lowest top of bank (see comment 1).
FCW(I)	r	0.1 - ∞	Set FCW(I) = channel width for rectangular channel. Set FCW(I) = width of channel base for trapezoidal channel.
FROUDC(K)	r	0.0 - 5	Maximum Froude number for channel flow. If the Froude number exceeds FROUDC, the Manning's n roughness value is increased by 0.001. Set FROUDC = 0 for no adjustments of the n-value in a given reach. The increased n-values are reported in the ROUGH.OUT and CHAN.RGH files (see comment 7).
ICHANGRID(I)	i	1 - NNOD	Grid element containing a channel length (see comments 3, 11, 14 and 20)
IEND	i	1 - NNOD	Last channel element for which a starting water surface elevation is specified.

Variable Descriptions for the CHAN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ISEDN(K)	i	0 - 10	Sediment transport equation or data group for routing by size fractions for the channel segment. Set ISED = 1 in the CONT.DAT file to use this option. Choose one of the two following options for each channel segment: For sediment routing without size fractions: Set ISEDN(K) = 1 - 7 (one of seven sediment transport equations). or For sediment routing with size fractions: Set ISEDN(K) = sediment data group (Line 3 in SED.DAT which includes a sediment transport equation).
ISTART	i	1 - NNOD	First channel element for which a starting water surface elevation is specified.
NOEXCHANGE	i	1 - NNOD	Channel element that does not exchange flow with the floodplain (see comment 19).
NXSECNUM(I)	i	1 to NNODC	Surveyed cross section number assigned in the XSEC.DAT file that will represent the specific channel element. This variable is used only for the cross section data option (see comments 14 and 18). Set NXSECNUM = 0, if there is no cross section data for the channel element (I). The cross section data will be interpolated and assigned in the PROFILES program.
ROUGHADJ	r	0.00 - 1.2	A coefficient used in the depth adjustment of the Manning's n-value (see comment 17).
SHAPE	c	R, V, T or N	Character line identifier (see comments 4 and 16); SHAPE = 'R', rectangular channel geometry (width and depth data). SHAPE = 'V', variable area channel geometry (power relationships). SHAPE = 'T', trapezoidal channel (bottom width, depth and slopes data). SHAPE = 'N', channel cross sections (cross section survey data);.
WSELEND	r	0 - 30,000 0 - 9,000	Ending water surface elevation for the channel element IEND (ft or m).
WSELSTART	r	0 - 30,000 0 - 9,000	Starting water surface elevation for the channel element ISTART (ft or m).

Variable Descriptions for the CHAN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
XLEN(I)	r	0.01 - ∞	Array of channel length contained within the grid element ICHANGRID (ft). If more than one channel exists in a given grid element, assign XLEN(I) equal to the average representative flow length in one direction (see comments 9, 10, 13 and 15).
ZL(I)	r	0.01 - 100	ZL(I) is the left side slope of the trapezoidal channel.
ZR(I)	r	0.01 - 100	ZR(I) is the right side slope of the trapezoidal channel.

Instructional Comments for the CHAN.DAT File

1. The channel bottom elevation is calculated by the model based on the input channel depth and the floodplain or bank elevation.

2. When DEPINITIAL > 0, an initial depth is specified for all the elements in that channel segment. Setting DEPINITIAL = -1 will assign starting and ending water surface elevations (WSELSTART and WSELEND, Line 3) for a channel segment beginning with channel element ISTART and ending with channel element IEND. Only one starting and ending water surface is allowed per channel segment. The water surface elevations are computed for the channel elements between the ISTART and IEND elements based on the interpolations of the channel length and the specified water surface elevations.

3. Dividing the channel into segments may simplify reviewing the results. Organize the CHAN.DAT from upstream to downstream. The order of the grid element numbers in the file is not important (e.g. upstream channel element 446 can precede downstream channel element 31). The channel grid elements must be contiguous in each segment.

4. If channel geometry is being simulated with regression relationships (SHAPE = 'V'), then the area versus depth power relationships must be specified:

$$A = ad^b$$

Where:

A = Area of the channel
d = Depth to thalweg
a = Coefficient
b = Exponent

Similar relationships are required for wetted perimeter and top width. There is a limit of two channel geometry relationships per channel element. A second geometry relationship may be useful if there is a significant change in the cross section (e.g. an island). If two power relationships are used to represent a natural cross section, then the maximum depth (EXCDEP) to which the first relationship applies must be specified.

The second regression applies when the flow depth is greater than EXCDEP, but does not include the lower flow area. The two variable area cross section relationships are unique and separate. The total cross section flow area is the sum of the lower flow and upper (second relationship) flow areas. The channel top width is computed directly from the second relationship. The area, wetted perimeter and top width are evaluated using the upper flow depth given

by total depth - EXCDEP. To analyze the upper channel geometry using the XSEC program, only the cross section coordinates above the EXCDEP depth are used.

These channel geometry relationships apply only to flow depths that are less than the channel depth (lower than the top of bank). When the flow depth exceeds the top of bank, then the channel geometry above bank is evaluated as a rectangle. Abrupt transitions between contiguous channel elements should be avoided unless they actually exist.

5. A preprocessor program XSEC is available in the FLO-2D subdirectory to determine the regression coefficient and exponents (A1, A2, A11, A22, B1, B2, B11, B22, B2, C1,C11, C22) in Line 2b.

6. A cross section width can exceed the width of the grid element. For example, a channel cross section that is 1,000 ft wide can be used in a 200 ft grid system. The model automatically determines the number of grid elements required by a channel cross section. If the cross section width exceeds 95% of the combined bank elements width or if there is less than 5% floodplain surface area left in the grid element after removing the channel surface area, the channel will extend the right bank over another grid element looking downstream.

7. Set the channel roughness to a reasonable n-value and then set the FROUDC variable to an appropriate value (e.g. 0.95 to ensure subcritical flow). FLO-2D will adjust the roughness values according to the limiting Froude number criteria (see the ROUGH.OUT file). Changes to the channel n-values may be accepted by replacing the CHAN.DAT file with the CHAN.RGH file. Just delete the original CHAN.DAT file and rename the CHAN.RGH to CHAN.DAT.

8. Channel confluence pairs (elements) replace NOFLOCs from previous versions. The confluence can be made by the tributary joinging either side of the main channel. List the tributary first and the main channel second in Line C.

9. Use the PROFILES program to review the channel slope and adjust the bed elevations to create a more uniform average channel reach slope. The PROFILES program can interpolate cross sections and slope for surveyed cross sections.

10. The key to channel routing is to balance the relationship between the slope, flow area and roughness. Channel routing is more stable if the natural cross section routing routine is used (SHAPE = N). When one cross section is assigned to several grid elements it will be necessary to interpolate both the slope and the cross section geometry in the PROFILES program to create a smooth average channel slope. Review the PROFILES program instructions for cross section and channel bed slope interpolation. If there is more than one cross section per channel element, use the one that has the greatest hydraulic control to represent the channel.

11. At a channel confluence, the next downstream channel element bed elevation must be lower than the confluence bed elevation creating a positive slope downstream of the confluence.

12. If different bank elevations are assigned, the model automatically extends the channel into separate grid elements, one grid element containing each bank. The model may be required to do this anyway because the channel is wider than the grid element.

13. The first two channel elements in a segment should have a positive slope in the downstream direction. This is important for inflow channel elements. There should also be a positive slope into the channel outflow nodes. This will enhance numerical stability around the inflow and outflow nodes.

14. If you delete a channel element, you must remove the cross section for that channel element from the XSEC.DAT file. If cross sections are mixed with other channel geometry, the cross section elements should be grouped into segments to identify the reaches with similar channel geometry.

15. Eliminate channel elements that have a XLEN less than 50% of the SIDE (grid element width). This can be accomplished by connecting the channel elements across the diagonal and eliminating the middle channel element.

16. For channel flow, adjust the full dynamic wave WAVEMAX in the TOLER. DAT to achieve numerical stability.

17. To improve the timing of the floodwave progression through the system, a depth variable roughness can be assigned on a reach basis. The basic equation for the channel element roughness n_d as function of flow depth is:

$$n_d = n_b \; a \; e^{-(b \; depth/dmax)}$$

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = bankfull flow depth
$a = 1/e^{-b}$
b = roughness adjustment coefficient prescribed by the user (0 to 1.2)

This equation prescribes that the variable depth channel roughness is equal to the roughness at bankfull discharge. If the user assigns a value (from 0 to 1.2) to the roughness adjustment coefficient b for a given reach, the roughness will increase with a decrease in flow depth. The higher the coefficient b, the greater the increase in roughness. This roughness adjustment will slow the progression of the floodwave by increasing the roughness for less than bankfull discharge. The plane bed roughness set for bankfull discharge will not be affected. For example, if the depth is 20% of the bankfull discharge and the roughness

adjustment coefficient b is set to 0.44, the hydraulic roughness Manning's n-value will be 1.4 times the roughness prescribed for bankfull flow. Assigning a ROUGHADJ value may reduce high Froude numbers.

18. Instructions for creating the cross section channel geometry data files. The FPLAIN.DAT, CADPTS.DAT, CONT.DAT and TOLER.DAT should be completed first.

    a. Create the XSEC.DAT file as prescribed in the next data file format description.

    b. Create the CHAN.DAT file. Assign the left bank by creating the channel segment using Create Channel Segment tool in the GDS. Then use the GDS to assign the right bank element. The GDS will automatically create the CHANBANK.DAT file containing the bank elements.

    c. Assign the cross section numbers (NXSECNUM in XSEC.DAT) to the channel elements in CHAN.DAT that corresponds to the cross section location. Assign NXSECNUM = 0 for the rest of the channel elements. If there are 100 channel elements and 30 surveyed cross sections in the XSEC. DAT file, there will be 30 NXSECNUM values (ranging from 1 to 30) in the CHAN.DAT file. The rest of NXSECNUM values will be zero.

    d. Once the CHAN.DAT and XSEC.DAT files are completed, the next step is to create and assign a unique cross section for each channel element. Run the PROFILES Interpolation Routine from the GDS. PROFILES will automatically locate the surveyed cross section data and interpolate the cross section geometry and elevation (thalweg slope) to all channel elements between all the surveyed cross sections with the segment. Now each channel element will have its own assigned cross section. By zooming-in on a local reach and clicking on menu command 'View/Edit Xsection Data' and then clicking on the XSEC button in the dialog box, the individual cross section plots and the data for editing will be displayed.

    e. After the original interpolation, you can further edit and interpolate the channel cross section data in PROFILES. When the initial interpolation was completed, the original cross sections were renamed with a prefix 'X-' before each cross section name. Either the slope or the combined slope and cross section geometry can be re-interpolated. Simply enter the number of the two channel elements with the upstream and downstream channel element numbers in the dialog box and click the appropriate interpolation button. New cross sections will be interpolated for all the channel elements between the two surveyed cross sections (exclusive). This is a flow area weighted interpolation of the cross section shape. The results will be an individually modified cross section for each channel grid element and the

channel profile will have a new linear slope between the two selected channel elements.

f. If there is a need to delete or add a channel element to a natural channel segment, the corresponding cross section must also be removed or created. This action should be done using a text editor such as WordPad©, TextPad© or UltraEdit©. To add a new channel element, simply copy the line above the location of the new element and paste it into a place holder for the new element. Then set the variable ICHANGRID(I) to the desired grid element number. If necessary alter the channel defining variables but do not alter NXSECNUM. Next, perform a similar procedure for the new cross section. Copy the cross section above the desired location of the new cross section. There should be no need to rename the cross section and do not alter NXSECNUM. Save and close the data files and open the project in PROFILES. Save the project in PROFILES to renumber the new channel element and its corresponding cross section. Then you can interpolate between the channel elements that border the new element and create a positive slope. You must follow the same procedure in PROFILES when you delete a channel element and cross section.

19. Channel elements representing a closed conduit or culvert that cannot share discharge with the floodplain can be simulated by assigning those elements as 'NOEXCHANGE'.

20. Surveyed water surfaces can be automatically compared with the predicted water surface by creating a WSTIME.DAT file. This file contains a list of the channel element, water surface elevation and time. Create this file to calibrate the model to known water surface elevation data. The time of the surveyed water surface elevation must correspond to the model flood routing timing.

FILE: CHANBANK.DAT

CHANNEL BANK DATA

CHANBANK.DAT File Variables

26 99	Line 1: **LEFTBANK(K)** **RIGHTBANK (K)**	*K = 1, number of channel elements*

Notes:
If ICHANNEL = 0 in the CONT.DAT file, omit this file.
Line 1: If a channel element width is contained within one grid element and no individual bank elements are assigned then **RIGHTBANK(K)** is set to zero.

CHANBANK.DAT File Example

```
26   99
39   136
54   156
71   176
90   196
109  216
127  236
147  256
167  276
187  315
207  336
226  356
247  377
267  398
286  418
307  439
327  460
348  481
369  502
```

Variable Descriptions for the CHANBANK.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
LEFTBANK	i	1 - NNOD	Left bank channel grid element. Assigned in CHAN.DAT as the ICHANGRID variable (Review the Instructional Comments).
RIGHTBANK	i	1 - NNOD	Right bank channel grid element corresponding the LEFTBANK element.

Instructional Comments for the CHANBANK.DAT File

1. The RIGHTBANK element is automatically assigned in the GDS. You can make adjustments to the right bank channel element if the channel is too wide or narrow by simply reassigning the right bank element in the GDS.
2. The procedure for assigning the right bank element is to first select the left bank element in the GDS, open the channel segment editor box and then assign the right bank element. The GDS will automatically check the channel width to determine if the channel bank assignments are appropriate and will report and required modifications in the ERROR.CHK file.
3. Channel right bank assignments are not required if the channel cross section will fit in one grid element and no bank elevations are assigned in CHAN.DAT.

FILE: XSEC.DAT

CROSS SECTION DATA

XSEC.DAT File Variables

X 1 X-CI-27.1	Line 1: **XSECCHAR = 'X' NXSECUM(I) XSECNAME(I) I=1**
25.0 5234.90	Line 2: **XI(I,J) YI(I,J)** *I = 1 Cross Sections J=1 Station Elevation pairs*
30.0 5231.53	Line 2: **XI(I,J) YI(I,J)** *I = 1 Cross Sections J=2 Station Elevation pairs*
35.0 5230.20	Line 2: **XI(I,J) YI(I,J)** *I = 1 Cross Sections J=3 Station Elevation pairs*
...	

Notes:
If ICHANNEL = 0 in the CONT.DAT file, omit this file.
Set SHAPE = 'N' (line 2d) in the CHAN.DAT file to use this file.
Line 1: This line is repeated for each cross section.
Line 2: This line is repeated for the Station, Elevation pairs.

XSEC.DAT File Example

```
X   1   X-CI-27.1
      0.0        5235.07
     10.0        5235.17
     25.0        5235.31
     30.0        5231.84
      ...          ...
      ...          ...
    288.0        5236.01
    294.0        5236.51
    313.0        5237.00
X   2     CI-27.1
     25.0        5234.90
     30.0        5231.53
     35.0        5230.20
     40.0        5228.50
     45.0        5227.20
     50.0        5224.35
     55.0        5224.73
     60.0        5226.85
      ...          ...
```

Variable Descriptions for the XSEC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
NXSECNUM(I)	i	1 to NNODC	Cross section number starting with 1 and ending with the last surveyed cross section. This number will be assigned to the channel element NXSECNUM in CHAN.DAT. See comment 1.
XI(I,J)	r	0.0 - ∞	Array of cross section station distances from the left end point (ft or m). The value of XI can be either positive or negative.
XSECHAR	c	X	Character 'X' (case sensitive) that identifies Line 1.
XSECNAME(I)	c	Alpha Numeric	Cross section name (less than 10 characters, not case sensitive). This name is for cross section ID purposes only, it is not used by the model.
YI(I,J)	r	0 - 30,000 0 - 9,000	Array of cross section elevations (ft or m) at each station. The value of YI can either positive or negative indicating elevations below sea level.

Instructional Comments for the XSEC.DAT File

1. The NXSECNUM in XSEC.DAT and CHAN.DAT must match and be listed in order from 1 to N number of natural channel elements. The natural channel elements in the CHAN.DAT file must start at 1 and continue in sequence to NNODC from the top of the file to the end.

FILE: HYSTRUC.DAT

HYDRAULIC STRUCTURE DATA

HYSTRUC.DAT File Variables

Line 1: **STRUCHAR = 'S' STRUCTNAME IFPORCHAN(I) ICURVTABLE(I) INFLONOD(I) OUTFLONOD(I) INOUTCONT(I) HEADREFEL(I) CLENGTH(I) CDIAMETER(I)** *I = number of structures*

S Patagonia 1 0 1713 1827 0 4425.23 0.0 0.0

Line 2: **STRUCHAR = 'C' HDEPEXC(I,J) COEFQ(I,J) EXPQ(I,J) COEFA(I,J) EXPA(I,J)** *I = number of structures, J = number of curves*

C 20.0 3.543 0.890

Line 3: **STRUCHAR = 'R' REPDEP(I,J) RQCOEFQ(I,J) RQEXP(I,J) RACOEF(I,J) RAEXP(I,J)** *I = number of structures, J = number of curves*

R 12.0 0.00 1.0

Line 4: **STRUCHAR = 'T' HDEPTH(I,J) QTABLE(I,J) ATABLE(I,J)** *I = number of structures, J = number of datasets in table*

T 0 0

Line 5: **STRUCHAR = 'F' TYPEC(I) TYPEEN(I) CULVERTN(I) KE(I) CUBASE(I)** *I = number of structures, Set ICURVTABLE = 2 in Line 1.*

F 1 2 0.040 0.1 0.0

Line 6: **STRUCHAR = 'D' ISTORMDOUT(I), STORMDMAXQ(I),** *I = number of drain structures.*

D 4 15

Notes:

If IHYDRSTRUCT = 0 in the CONT.DAT file, omit this file.
Line 2: Include this line for rating curve. Repeat this line for each rating curve.
Line 1, 2: If CLENGTH(I) = 0, ignore COEFA(I,J) AND EXPA(I,J)
Line 3: If a replacement rating curve is required, include this line.
Line 1, 3: If CLENGTH(I) = 0, ignore RACOEF(I,J) and RAEXP(I,J).
Line 5: If TYPEC(I) = 2 (round pipe), CUBASE(I) = 0,
Line 4: If a rating table is used, include this line. Repeat for each depth and flowrate pair.
Line 1, 4: If CLENGTH(I) = 0, ignore ATABLE(I,J).

FILE: HYSTRUC.DAT

HYDRAULIC STRUCTURE DATA

HYSTRUC.DAT File Example

```
S  BridgeA  1  0  1713  1827  0   4425.23  0.0  0.0
C  20.0  3.543  0.890
S  BridgeB  0  0  2503  2725  1  0.0  0.0  0.0
C  5.0    25.023   1.035
C  10.0  30.00    1.4
R  12.0   0.00    1.0
S  Wier  1  1  1856  1945  0  4421.18  0.0  0.0
T  0.0        0.0
T  5.0      250.0
T  8.0     5500.0
T  10.0   1000.0
T  12.5   1500.0
T  15.0   2500.0
T  17.0   3000.0
S CulvertA  1  2  4417   4562  0  0.0   100.   90.
F  1  2  0.004   0.1   0.0
```

Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ATABLE (I,J)	r	0.01 - ∞	This variable will replace QTABLE(I,J) when CLENGTH(I,J) > 1 and the long culvert routine is used. It is an array of hydraulic structure flow areas for the headwater depths in the rating table (discharge).
COEFA(I,J)	r	0 - ∞	This variable will replace COEFQ(I,J) when CLENGTH(I,J) > 1 and the long culvert routine is used. It is an array of hydraulic structure flow area rating curve coefficients where the flow area A is expressed as a power function of the headwater depth. A = COEFA(I,J) * $depth^{EXPA(I,J)}$. See comment 7.
COEFQ(I,J)	r	0 - ∞	Array of hydraulic structure discharge rating curve coefficients where the discharge is expressed as a power function of the headwater depth. Q = COEFQ(I,J) * $depth^{EXPQ(I,J)}$ (see comment 1). If COEFQ(I,J) = 0, then the discharge is computed as normal depth flow routing.
CDIAMETER(I,J)	r	0.1 - ∞	Circular culvert diameter (ft or m). For the generalized box culvert equations CDIAMETER is the circular culvert diameter or the box culvert height. Also use CDIAMETER when a routing flow through a long culvert between two non-contiguous grid elements. See comment 13
CLENGTH(I,J)	r	50.0 - ∞ 15.0 - ∞	Culvert or conduit length (ft or m). This parameter is used when a long culvert or conduit is simulated between two non-contiguous grid elements. CLENGTH must be assigned a value greater than 1 to initiate the long culvert routine.
CUBASE(I)	r	0 - ∞	Flow width of box culvert for TYPEC(I) = 1. For a circular culvert, CUBASE = 0.
CULVERTN(I)	r	0.012 - 0.25	Culvert Manning's roughness coefficient. Default = 0.03.
EXPA(I,J)	r	0 - ∞	This variable will replace EXPQ(I,J) when CLENGTH(I,J) > 1 and the long culvert routine is used. It is an array of hydraulic structure flow area exponents where the flow area is expressed as a power function of the headwater depth.
EXPQ(I,J)	r	0 - ∞	Array of hydraulic structure discharge exponents where the discharge is expressed as a power function of the headwater depth.
HDEPEXC (I,J)	r	.01 - 1000 0.25 - 300	Maximum depth that a hydraulic structure rating curve is valid (ft or m).
HDEPTH(I,J)	r	.01 - 1000 0.25 - 300	Array of headwater depths for the structure headwater depth-discharge rating table (ft or m). See comments 2 and 4.

Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
HEADREFEL(I)	r	.01 - 30,000 .25 - 9,000	Reference elevation above which the headwater depth is determined for either the discharge rating curve or rating table. Set HEADREFEL(I) = 0.0 to use the existing channel bed or floodplain elevation for the reference elevation to compute the headwater depth (ft or m).
ICURVTABLE(I)	s	0 = curve 1 = table	Set ICURVTABLE(I) = 0 for a structure rating curve. Set ICURVTABLE(I) = 1 for a structure rating table. Assign ICULVTABLE(I) 2 to use the culvert equations. See comment 6.
IFPORCHAN(I)	s	0, 1 or 2	Set IFPORCHAN(I) = 0 for a floodplain structure (shares discharge between two floodplain elements). Set IFPORCHAN(I) = 1 for a channel hydraulic structure (shares discharge between to channel elements). Set IFPORCHAN(I) = 2 for a floodplain to channel structure (shares discharge between a floodplain element {inflow} and channel structure {outflow}). See comment 8.
INFLONOD(I)	i	1 - NNOD	Grid element containing the hydraulic structure or structure inlet.
INOUTCONT(I,J)	s	0 = inlet 1 = revised 2 = outlet / revised	Set INOUTCONT(I,J) = 0 to compute the discharge based on the headwater depth above the appropriate floodplain or channel bed elevation (or reference elevation if assigned). No revised rating table, but suggested revisions in REVISED_RATING_TABLE.OUT. This can be used for pumps. No tail water. Set INOUTCONT(I,J) = 1 revised rating table, no tailwater, no upstream flow, table revised automatically, revisions posted to REVISED_RATING_TABLE.OUT. Set INOUTCONT(I,J) = 2 revised rating table, upstream flow possible, tail water submergence effective, revisions posted to REVISED_RATING_TABLE.OUT.
ISTORMDOUT(I)	i	1 - NNOD	Grid element number that is one of the hydraulic structure outflow elements to simulate a simplified storm drain. ISTORMDOUT is a junction or outflow node for a number of inflow nodes. See comment 12.
KE(I)	r	0.01 - 1.0	Culvert entrance loss coefficient. See comment 10.

Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
OUTFLONOD(I)	i	1 - NNOD	Grid element receiving the hydraulic structure discharge (structure outlet). OUTFLONOD does not have to be contiguous to INFLONOD grid element.
QTABLE(I,J)	r	0.01 - ∞	Array of hydraulic structure discharges for the headwater depths in the rating table (discharge). See comments 3 and 5.
REPDEP(I,J)	r	0.01 - ∞	Flow depth (ft or m) that if exceeded will invoke the replacement structure rating curve parameters for simulating a blockage or a change in the rating curve.
RACOEF(IJ)	r	0 - ∞	This variable will replace RQCOEF(I,J) when CLENGTH(I,J) > 1 and the long culvert routine is used. It is an array of structure rating curve discharge replacement coefficients. There should be the same number of rating curve pairs of coefficients and exponents.
RAEXP(IJ)	r	0 - ∞	This variable will replace RQEXP(I,J) when CLENGTH(I,J) > 1 and the long culvert routine is used. It is an array of structure rating curve discharge replacement exponents. There should be the same number of rating curve pairs of coefficients and exponents.
RQCOEF(I,J)	r	0 - ∞	Structure rating curve discharge replacement coefficients. There should be the same number of rating curve pairs of coefficients and exponents
RQEXP(I,J)	r	0 - ∞	Structure rating curve discharge replacement exponents. There should be the same number of rating curve pairs of coefficients and exponents.
STRUCHAR	c	S, C, R, T or D	Character that identifies the use of Line 2, 3, 4 or 6 where: STRUCHAR = 'S' for the structure control, (Line 1); STRUCHAR = 'C' for a rating curve (Line 2); STRUCHAR = 'R' for replacement rating curve (Line 3); STRUCHAR = 'T' for a rating table (Line 4); STRUCHAR = 'F' for culvert equations (Line 5); STRUCHAR = 'D' for storm drain (Line 6).
STORMDMAXQ(I)	r	0 - ∞	Maximum allowable discharge (conveyance capacity) of the collection pipe represented by the ISTORMDOUT element.
STRUCTNAME(I)	i	Alpha Numeric	Hydraulic structure name (15 characters or less). This name is for user identification purposes only. No Spaces allowed in the name.

Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
TYPEC(I)	s	1 = box 2 = pipe	Culvert switch, either 1 or 2. Set TYPEC(I) = 1 for a box culvert and TYPEC(I) = 2 for a pipe culvert. See comment 9.
TYPEEN(I)	s	1, 2, 3	Culvert switch. Set TYPEEN(I) for entrance type 1, 2, or 3. See comment 9.

Instructional Comments for the HYSTRUC.DAT File

1. Hydraulic structure discharge between either floodplain or channel grid elements can be simulated as a rating curve function of the headwater depth, Q = COEFQ*depth^{EXPQ}, where COEFQ and EXPQ are specified coefficients and exponents which are valid for a depth not to exceed HDEPEXC. The grid elements containing the structure inlet and outlets must be specified. The inlet and outlet grid elements do not have to be contiguous. The structure discharge (such as a culvert, weir or bridge) may either inlet or outlet control as long as the discharge is specified as power function of the headwater depth.

2. When the headwater depth exceeds the specified depth (HDEPEXC) for which the original rating curve relationship is valid, a second replacement relationship is invoked. These multiple relationships can be used to specify structure blockage or a change in the rating curve. For example, if a height of 5 ft corresponds to the top of a culvert for a discharge of up to 300 cfs, then a second rating curve relationship for flows over a roadway could be based on a flow depths starting at 6 ft above the culvert invert that could correspond to a discharge of greater than 500 cfs. Structure blockage can be simulated by setting the replacement coefficient (RQCOEF) equal to zero.

3. If a hydraulic structure rating table is used, a linear interpolation between two headwater depths in the table is applied to estimate the discharge for a headwater depth determined by the model.

4. When structure tailwater effects must be considered, the HDEPEXC variable for the rating curves represents the difference between the headwater and tailwater water surfaces.

5. The rating table should always have the first pair of depth-discharge data as headwater depth = 0 and discharge = 0 to enable interpolation with the next data pair in the rating table.

6. The hydraulic structure may be any type of flow control such as a bridge, diversion, culvert, weir, roadway or spillway. If a short culvert is simulated that is separated by more than one grid element, neither the travel time or volume of storage in the culvert is considered. The discharge is computed at the outflow element for the same timestep. This is a relatively minor assumption that should not affect your simulation unless the culvert can contain a significant portion of the flood volume in the entire model.

7. If the culvert or conduit is long, an alternate set of equations will be invoked. These equations are governed by the variables COEFA, EXPA, RACOEF,

RAEXP and ATABLE. The model will use automatically substitute the long variables and equations when CLENGTH > 1.

8. If the hydraulic structure is a culvert between two floodplain elements, set IFPORCHAN = 0. If the structure is a bridge or culvert in a channel, set IFPORCHAN = 1. If the structure such as a culvert collects discharge from a floodplain and discharges to a channel, set IFPORCHAN = 2.

9. The Department of Transportation generalized culvert equations can be used to assess inlet and outlet control. The type of culvert entrances are:.

   BOX entrance:

   type 1 - wingwall flare 30 to 75 degrees
   type 2 - wingwall flare 90 or 15 degrees
   type 3 - wingwall flare 0 degrees

   PIPE entrance:

   type 1 - square edge with headwall
   type 2 - socket end with headwall
   type 3 - socket end projecting

10. The culvert equations use the conventional KE values that be found in the literature.

11. If INOUTCONT(I,J) = 0, then the hydraulic structure discharge is based solely on the upstream water surface elevation or headwater depth on the reference elevation (assigned or node elevation). This is equivalent to inlet control for a culvert. If INOUTCONT(I,J) = 1, then the tailwater submergence and potential flow reversal (from downstream to upstream) is evaluated. As the tailwater elevation approaches the upstream headwater elevation, the model adjusts the rating curve and reduces the discharge gradually. If the tailwater elevation exceeds the headwater elevation then flow upstream is possible. When the switch INOUTCONT(I,J) = 1, if the hydraulic structure discharge is greater than the upstream inflow, the headwater elevation decreases and the tailwater elevation increases. As the two water surface elevations on each side of the structure equilibrate the submergence factor and the structure discharge are reduced. This may occur because of tailwater effects or because the structure discharge rating table was overestimated for the upstream flow conditions. The modifications to the rating table are reported in the file REVISED_RATING_TABLES.OUT file.

12. By assigning ISTORMDOUT, the discharge from this outflow element will represent the collective inflow from any number of upstream inflow discharge from using the hydraulic rating curve or table. The discharge in the outflow element ISTORMDOUT can be limited to the maximum discharge value STORMDMAXQ. When the STORMDMAXQ is exceeded, no additional inflow discharge will be computed for successive downstream inflow nodes.

This simplified storm drain routine does not include any pipe flow routing and does not use the SWMM model. The purpose of this component is to estimate the collected discharge in a large storm drain pipe and limit the potential inflow to the pipe capacity without performing a pipe network discharge calculation. For complex pipe networks, use the FLO-2D SWMM model interface.

13. CDIAMETER is primarily used to estimate the timing of flow through a long culvert. This is accomplished with a Muskingcum type storage routing. When the culvert is longer than about 1,000 (300 m), the timing of the flow in the culvert may not match the timing of the floodwave progression. Generally, the amount of storage in the culvert is not significant compared to the flood volume. Use CDIAMETER for a box culvert width if the generalized culvert equations are used. When using other culvert shapes such as an oval, define an approximate equivalent circular culvert diameter. For multiple box culverts, define an equivalent single box culvert width (CuBASE) and height (CDIAMETER).

FILE: STREET.DAT

STREET DATA

STREET.DAT File Variables

0.025 1 1.7 0.667 40	Line 1:	**STRMAN ISTRFLO STRFNO DEPX WIDST**
N MAIN	Line 2:	**STCHAR = 'N' STNAME**
S 127 0 0 0	Line 3:	**STRCHAR = 'S' IGRIDN(L) DEPX(L) STMAN(L) ELSTR(L)**
W 1 40	Line 4:	**STRCHAR = 'w', ISTDIR(K) WIDR(K)** *K = 1,8 street elements*
W 2 50	Line 4:	**STRCHAR = 'w', ISTDIR(K) WIDR(K)** *K = 1,8 street elements*
W 4 50	Line 4:	**STRCHAR = 'w', ISTDIR(K) WIDR(K)** *K = 1,8 street elements*
S 128 0 0 0	Line 3:	**STRCHAR = 'S' IGRIDN(L) DEPX(L) STMAN(L) ELSTR(L)**
		L = number of grid elements in each street segment.

Notes:
If MSTREET = 0 in the CONT.DAT file, omit this file.
If DEPEX, STMAN, ELSTR, and WDIR = 0 the global values from Line 1 will be used.
Each grid element should be listed only once in this file.
Line 2 - 4: Repeat these lines for each street.
Line 4: Repeat this line for the number of grid elements before repeating Line 3.

STREET.DAT File Example

```
0.025     1    1.7  0.667     40
N  MAIN
S127      0     0     0
W         1    40
W         2    50
W         4    50
S       128     0     0     0
W         2    50
W         4    50
S       129     0     0     0
W         2    50
W         4    50
S       131     0     0     0
W         2    50
```

Variable Descriptions for the STREET.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DEPX	r	0.0 - 2.0 0.0 - 0.6	Street curb height (ft or m). If the street curb height is exceeded by the flow it will result in overland flow in the grid element containing the street. DEPX is used to assign a street curb height to all grid elements (see comment 7).
DEPEX(L)	r	0.01 - 2 0.25 - .6	Optional curb height (ft or m) for individual grid elements that supercedes the global curb height DEPX. Set DEPEX(L) = 0.0 to use DEPX.
ELSTR(L)	r	0 - 30,000 0 - 9,000	Optional street elevation (ft or m). This elevation will supercede the floodplain grid element elevation. If ELSTR(L) = 0, the model will assign the street elevation as grid element elevation, FP(I,6) minus the curb height DEPEX(L) or DEPX to the street elevation ELSTR(L) (see comment 3).
IGRIDN(L)	i	1 - NNOD	Grid element number. Each grid element should be listed only once in the data file (see comment 6).
ISTDIR(K)	I	1 - 8	Street segment (flow direction) from the center of the grid element to a neighboring element. IITDIR(k) will vary from 1 to 8 according to the following compass directions: 1 = north 5 = northeast 2 = east 6 = southeast 3 = south 7 = southwest 4 = west 8 = northwest
ISTRFLO	s	0 or 1	ISTRFLO = 1 specifies that the system inflow hydrograph will enter the streets rather than entering the overland portion of the grid element.
STRCHAR	c	N, S or W	Character 'N', 'S' or 'W' to identify either Line 2, 3 or 4.
STRFNO	r	0.0 - 5	Maximum street Froude number. When the computed Froude number for the street flow exceeds STRFNO, the n-value is increased by 0.001 for that grid node. The increased n-values are reported in the ROUGH.OUT and STREET.RGH files
STMAN(L)	r	0.01 - 0.25	Optional street n-value for the individual street segments within a given grid element. STMAN(L) supercedes the STRMAN value. If STMAN(L) = 0.0, the global value STRMAN will be assigned to the grid element street segment.
STRMAN	r	0.01 - 0.25	Global n-value for street flow which that is assigned to all the grid element street segments (See comment 2).

Variable Descriptions for the STREET.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
STNAME	c	Alpha Numeric	Character name of the street. Up to 15 characters can be used. The street name is not used in the model (see comment 1).
WIDR(K)	r	0.0 - 1,000 0.0 - 300	Optional grid element street width in the ISTDIR direction. If the grid element contains more than one street, Line 4 must be repeated. If a given grid element has more than one street in one direction, modify WIDR(K) to represent the combined widths of the streets. Up to 8 street segments, one for each of the 8 compass directions, can be assigned according to the ISTDIR variable. By setting WIDR(K) = 0.0, the WIDST global width will be assigned to that street segment (see comments 4 and 5).
WIDST	r	0.01 - ∞	Global assignment of street width to all streets. This value is superseded by WIDR(K) when WIDR(K) is greater than zero (see comments 2 and 4).

Instructional Comments for the STREET.DAT FILE

1. The street name is provided for the user to separate the streets groups for easy identification in the data file. It is not used in the program.

2. The street depth, width and n-values can be assigned globally for all the street grid elements. The street depth, width, n-value and elevation can be uniquely assigned for individual grid elements in Lines 3 and 4 by setting these values greater than zero.

3. If the street elevation is different from the representative grid elevation assigned in the FPLAIN.DAT file, it should be specified in line 3, otherwise the street elevation will be the floodplain elevation minus the curb height. This elevation is then used to determine the street slope.

4. The street width should be less than the width of the grid element. The overall floodplain surface area of the grid after the streets are removed must be at least 5% of the original surface area (grid element width squared). If there are numerous streets in the grid element that consume all the grid element surface area, consider leaving out the smaller, less significant streets, reduce the street width or transfer one or more to a neighboring grid element. Another option is to increase the grid element size and redo the grid system.

5. The street is assumed to extend from the center of the grid element to the grid element boundary in the four compass directions plus the four diagonal directions as specified by the variable ISTDIR(K). A street that crosses the entire grid element is assigned two street sections and directions in Line 4.

6. Each grid element should be listed only once in the STREET.DAT file. For street intersections within the grid element, list all the street flow directions for the first street, then skip that grid element for the succeeding crossing streets.

7. The street flow depth TOL value below which no computations are perform is 0.03 ft or 0.01 meters. This value is comparable to TOL value in TOLER.DAT but it is hardwired into the model. The user cannot adjust it.

FILE: ARF.DAT

FLOODPLAIN AREA AND WIDTH REDUCTION DATA

ARF.DAT File Variables

S 0	Line 1:	**ITTCHAR = 'S' ARFBLOCKMOD**
T 49	Line 1:	**ITTCHAR = 'T' ITTAWF(K)**
29 .2 .70 .50 1.0 0. 0. 0. 0. 0.	Line 2:	**IDG(I) ARF(I) WRF(I,J)**
		K = number of totally blocked grid elements
		I = number of partially blocked grid elements
		J = 8 flow directions

Notes:
If IWRFS = 0 in the CONT.DAT file, omit this file.
Line 1: Repeat this line for each totally blocked grid element.
Line 2: Repeat this line for each partially blocked grid element.

ARF.DAT File Example

```
S 0.
T 540
T 2502
T 3818
T 3861
T 4435
T 4766
 46 .1 0 .5 0 .5 0 0 0 0
 69 .3 0 0 0 0 0 0 0 0
 119 .4 .5 .7 1 0 0 0 0 0
 120 0 0 0 0 1 0 .2 0 0
 142 .2 .2 0 0 0 0 0 0 0
 161 .5 0 0 0 0 0 0 0 0
 162 .5 .7 .2 1 0 0 0 0 1
 163 .1 0 0 0 1 0 0 0 0
 182 .3 0 0 0 0 0 0 .3 0
 ....
```

Variable Descriptions for the AFR.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ARF(I)	r	0 - 1	Array of optional area reduction factors (ARF) for grid element I. ARF(I) is the percent of the grid element area that can not be covered by surface flows. Buildings or other physical features contained within the grid that cannot store flow volume are accounted by using the ARF value. The maximum value for an ARF = 0.95. If ARF > 0.95, then ARF is reset to 1 (see comments 1 and 3).
IARFBLOCKMOD	r	0. - 1.	Global revision to the ARF = 1 value to the grid elements that are total blocked from receiving any flow (ITTAWF elements). Setting IARFBLOCKMOD = 0.9 will change the ARF = 1. to ARF = 0.9 for all the ITTAWF elements. See comment 4.
IGD(I)	i	1 - NNOD	Array of partially blocked grid element numbers.
ITTAWF(I)	i	1 - NNOD	Array of grid elements that will not receive any flow. Each grid element is totally blocked out and all ARF and WRF values are set equal to 1.0.
ITTCHAR	c	T	Set ITTCHAR = 'T' to identify Line 1.
WRF(I,J)	r	0 - 1	Array of optional width reduction factors (WRF) for grid element(I). The width reduction factor corresponds to the percentage of flow width lost due to obstruction in four directions, north, east, northeast and southeast. The width reduction for the other four directions is defined by the values assigned to the contiguous elements. Assuming that the flow field is oriented with the north direction toward the top of the page, use the following WRF assignment: WRF(I,1) = North WRF(I,2) = East WRF(I,3) = South WRF(I,4) = West WRF(I,5) = Northeast WRF(I,6) = Southeast WRF(I,7) = Southwest WRF(I,8) = Northwest where I is the grid element number (see comment 2).

Instructional Comments for the ARF.DAT File

1. For a partially blocked grid element, those ARF and WRF values that are 0.0 must be entered. The graphical assignment and editing of ARF and WRF values are easy in the GDS program.

2. Each grid element can receive or discharge flow through eight sides. Consider each element to be an octagon. Each WRF factor refers to the percent blockage of one of the eight sides.

3. The maximum ARF value is 0.95 unless the grid element is totally blocked out in Line 1 as a ITTAWF grid element. This insures that at least 5% of the grid element is left for flow storage. For ARF values greater than 0.95, the model will reset the ARF to 1. Use the ITTAWF parameter to totally block out a grid element with zero storage.

4. Instead of completely blocking any flow from entering the ITTAWF elements, assigning IARFBLOCKMOD < 1. will allow some flow storage in these completely blocked elements.

FILE: MULT.DAT

MULTIPLE CHANNEL (RILL AND GULLY) DATA

MULT.DAT File Variables

15 1 1 0.060 0.005 0.010	Line 1:	**WMC WDRALL DMALL NODCHNSALL XNMULTALL SSLOPEMIN, SSLOPEMAX**
13 .50 6 1 0.055	Line 2:	**IGRID(I) WDR(I) DM(I) NODCHNS(I) XNMULT(I)** *I = number of grid elements with multiple channels*

Notes:
If MULTC = 0 in the CONT.DAT file, omit this file.
If WDRALL = 0, no global assignment of the variables occurs.
Line 3: Repeat this line for each grid element revision.

MULT.DAT File Example

15	1	1	0.060	0.005	0.009
2	50	6	1	0.055	
27	20	3	1	0.06	
28	20	3	1	0.06	
29	20	3	1	0.06	
30	20	3	1	0.06	
50	10	2	1	0.06	
51	10	2	1	0.06	
52	10	2	1	0.06	
53	10	2	1	0.06	
54	10	2	1	0.06	
55	10	2	1	0.06	
56	10	2	1	0.06	
57	10	2	1	0.06	
58	10	2	1	0.06	
79	5	1	1	0.06	
80	5	1	1	0.06	
81	5	1	1	0.06	

Variable Descriptions for the MULT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DM(K)	r	0 - 1000 0 - 300	Maximum depth of multiple channels for individual grid elements (ft or m). When the flow depth exceeds the multiple channel depth DM, the flow width WDR of the gully is increased by the incremental width WMC (see comments 2 and 3). DM supercedes the DMALL depth assignment.
DMALL	r	0 - 1000 0 - 300	Global assignment of the maximum depth to all grid elements (ft or m).
IGRID(I)	i	1 - NNOD	Floodplain grid element number (see comment 1)
NODCHNS(K)	i	0 - 100	Number of multiple channels assigned in each grid element. If NODCHNS is set equal to zero then the overland flow without multiple channels is assumed. NODCHNS supercedes NODCHNSALL value.
NODCHNSALL	i	1 - 100	Global assignment of the number of multiple channels to all grid elements.
SSLOPEMIN	r	0. - 1.	Minimum slope that multiple channel assignments will be made at runtime.
SSLOPEMAX	r	0. - 1.	Maximum slope that multiple channel assignments will be made at runtime.
WDR(K)	r	0 - 1000 0 - 300	Channel width for individual grid elements. WDR supercedes WDRALL.
WDRALL	r	0 - 1000 0 - 300	Global assignment of the multiple channel width to all grid elements. If WDRALL = 0, all global variables are set to zero.
WMC	r	0 - 1000 0 - 300	Incremental width by which multiple channels will be expanded when the maximum rill and gully depth DM is exceeded (see comments 2 and 4).
XNMULT(K)	r	0.01 - 0.5	Channel n-values for individual grid elements. Supercedes XNMULTALL.
XNMULTALL(K)	r	0.01 - 0.5	Global assignment of the multiple channel n-values to all the grid elements.

Instructional Comments for the MULT.DAT File

1. If a grid element is assigned multiple channels and, in addition, contains a main channel or buildings such that the available floodplain surface storage area is less than 50% of the original grid element surface area, then the model will reset that grid element to overland sheet flow (i.e. no multiple channels). Multiple channels will not be simulated in the grid elements with streets. The program will automatically eliminate any multiple channels in grid elements with streets. The user does not have to alter the data files. The available surface area and the assigned variable can be reviewed in the SURFAREA.OUT output file.

2. If a gully channel fills and is about to overflow, it is assumed that the gully is an alluvial channel and will widen to accept more flow. Thus when the flow depth exceeds the maximum gully channel depth DM, the model increases the gully width by the incremental width value WMC. Once the flow is in a gully, the gully will not overflow back onto the overland area, but will continue to widen until the gully is wider than the grid element or until there is no more overland storage area left in the grid element. The flood routing will then revert to overland flow in that element. The following rules govern the assignment of the multiple channel data:

   - When the flow depth exceeds the multiple channel (gully) depth, the flow width of the gully is increased by the incremental width.
   - If it is desired to force the flow to stay in a channel of fixed width, set the incremental width equal to zero.
   - if the number of multiple channels assigned in a grid element is set equal to zero, overland sheet flow without multiple channels is assumed.
   - The values given in the single grid element data group will supersede those in the global parameter group.

3. If it is desired to force the flow to stay in a channel of fixed width, set the variable WMC = 0.

4. The total flow width is determined by multiplying the number of channels in each grid element by the appropriate width.

5. SSLOPEMIN and SSLOPEMAX define a range of watershed slope in which the multiple channel width will be expanded. This will limit the rill and gully growth to the middle portion of the basin and will not expand the main channel if it is being modeled as a multiple channel. By expanding the rills and gullies for increased conveyance capacity, the time of concentration can be reduced. The default SSLOPEMIN = SSLOPEMAX = 0. will result in width increases for all multiple channels.

FILE: SED.DAT

MUDFLOW AND SEDIMENT TRANSPORT DATA

SED.DAT File Variables

Line 1: **SEDCHAR = 'M' VA VB YSA YSB SGSM XKX**

M 0.000602 33.10 0.001720 29.50 2.74 0.00

Line 2: **SEDCHAR = 'C' ISEDEQG ISEDSIZEFRAC DFIFTY SGRAD SGST DRYSPWT CVFG ISEDSUPPLY ISEDISPLAY**

C 2 0.25 2.5 2.65 92.5 1 7232

Z 2 5.0 0.15	Line 3:	**SEDCHAR = 'Z' ISEDEQI BEDTHICK CVFI**
P 0.062 0.010	Line 4:	**SEDCHAR = 'P' SEDIAM SEDPERCENT**
D 111 20.0	Line 5:	**SEDCHAR = 'D' JDEBNOD DEBRISV**
E 1.0	Line 6:	**SEDCHAR = 'E' SCOURDEP**
R 9366	Line 7:	**SEDCHAR = 'R' ICRETIN(N)** *N = number of rigid bed nodes*
S 23798 1 4.49 0.89	Line 8:	**SEDCHAR = 'S' ISEDGRID(N) ISEDCFP(N) ASED(N) BSED(N)** *N = number of sediment supply rating curves.*
N 0.062 0.052	Line 9:	**SEDCHAR = 'N' SSEDIAM SSEDPERCENT**
Z 1 3	Line 10:	**SEDCHAR = 'G' ISEDUM ISEDGROUP(N)** *N = number of sediment groups*

Notes:

Only ISED or MUD can be simulated at one time.

If MUD = 0 in the CONT.DAT file, omit line 1.

If ISED = 0 in the CONT.DAT file, omit line 2, 3, 4, 6, 7, 8, and 9.

If both MUD and ISED = zero in the CONT.DAT file, omit this file.

Line 2: If ISEDSIZEFRAC = 1, it is necessary to create a sediment group using Lines 3 and 4.

Line 4: Repeat this line for each size fraction. Each group must have the same number of size fractions.

Line 5: If IDEBRV = 0 in the CONT.DAT file, ignore this line.

Line 8, 9: If ISEDSUPPLY = 0, ignore these lines.

FILE: SED.DAT

MUDFLOW AND SEDIMENT TRANSPORT DATA

SED.DAT File Example

```
M  0.000602  33.10  0.001720  29.50  2.74   0.00  (Mudflow)
or
C 2 1 2.5 6.7 2.65 95.0 0.10 0 1961   (Sediment Transport)
Z 2 1. 0.10
P  0.074  0.058
P  0.149  0.099
P  0.297  0.156
P  0.590  0.230
P  1.19   0.336
P  2.38   0.492
P  4.76   0.693
P  9.53   0.808
P  19.05  0.913
P  38.10  1.000
E 1.0
R 2062
R 2063
R 2114
R 2115
R 2166
R 2167
S 1228 1  4.49  0.89
N 0.074 0.022
N 0.300 0.107
N 0.600 0.232
N 2.000 0.528
N 4.750 0.748
N 9.530 0.852
N 19.050 0.926
N 38.100 0.973
N 76.200 1.000
G  1  2
G  2  2
G  3  1
G  4  2...
```

Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ASED(N)	r	0 - ∞	Sediment rating curve coefficient (see the BSED variable below).
BEDTHICK	r	0 - 100 0 - 30	Sediment bed thickness (ft or m) for sediment routing by size fraction. The available sediment volume for a size fraction within a grid element is defined by the bed thickness times the floodplain or channel element surface area times the percent size distribution. The default bed thickness is 10 ft (3 m) for the floodplain if bed thickness is less than 0.1 ft. If there is no available sediment volume for a given size fraction, no further scour of the bed will occur for that sediment size fraction (see comment 2).
BSED(N)	r	0 - ∞	Sediment rating curve exponent. $Q_s = ASED * Q_w^{BSED}$ where: Q_w is the water discharge (cfs or cms) Q_s is the sediment supply (tons/day or kg/day).
CVFG	r	0 - 0.2	Fine sediment volumetric concentration for overland, channel, and streets. This value is superceded by CVFI in Line 3. Concentration by volume of sediment for sizes less than 0.0625 mm (sand-silt split). This concentration by volume generally ranges from 5% to 15% and is expressed as a decimal (0.05 for 5% concentration by volume). It is used only in Woo-MPM sediment transport equation.
CVFI	r	0 - 0.2	This variable is the same as CVFG except that it represents the fine sediment volumetric concentration for an individual channel segment(s). CVFI supercedes CVFG for a channel segment(s) as identified by ISEDN in CHAN.DAT. CVFI represents the concentration by volume of sediment for sizes less than 0.0625 mm (sand-silt split). This concentration by volume generally ranges from 5% to 15% and is expressed as a decimal (0.05 for 5% concentration by volume).
DEBRISV	r	0 - ∞	Volume of the debris basin in ft^3 or m^3.
DFIFTY	r	30625 - ∞	Sediment size (D_{50}) in mm for sediment routing.
DRYSPWT	r	70 - 130	Dry specific weight of the sediment (lb/ft^3 or N/m^3).
ICRETN	i	1 - NNOD	Floodplain or channel grid elements with a rigid bed (e.g. spillway apron).
ISEDCFP(N)	s	0 = fp 1 = chan	ISEDCFP(N) = 0 for a floodplain sediment supply rating curve ISEDCFP(N) = 1 for a channel sediment supply rating curve.

Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ISEDEQG	i	1 - 11	Transport equation number used in sediment routing for overland flow, channels and streets (see comment 3). In Line 2 (Line 'C'), ISEDEQG will set the sediment transport equation for floodplain sediment routing and channel routing. In Line 3 (Line 'Z'), ISEDEQI will set the sediment transport equation for sediment routing by size fractions with a sediment transport equation assigned to each group. Set ISEDEQG or ISEDEQI as follows for the appropriate sediment transport equation: ISEDEQ = 1 Zeller and Fullerton ISEDEQ = 2 Yang ISEDEQ = 3 Englund and Hansen ISEDEQ = 4 Ackers and White ISEDEQ = 5 Laursen ISEDEQ = 6 Toffaleti ISEDEQ = 7 Woo-MPM ISEDEQ = 8 MPM-Smart ISEDEQ = 9 Karim-Kennedy ISEDEQ = 10 Parker, Klingeman & McLean ISEDEQ = 11 Van Rijn
ISEDEQI	i	1 - 11	This variable is the same as ISEDEQG except that it represents the sediment transport equation used for sediment routing by size fractions and to identify the sediment transport equation that will be used by a specific channel segment or reach (comment 5). This value supercedes ISEDEQG in Line 2. In Line 3 (Line 'Z'), ISEDEQ will set the sediment transport equation for sediment routing by size fractions with a sediment transport equation assigned to each group. If Line 3 and the following Line 4's constitute only one group, then all sediment routing on the floodplain, in the channel and in the streets will use the same sediment size distribution. If there is more than one group of Line 3 and the following Line 4's, then the first group will be define the sediment size distribution for the floodplain, streets and any channel segments where ISEDN = 1 in CHAN.DAT. Successive channel segments can identify another set of sediment size fractions by setting ISEDN = 2 or higher. This will permit the channel bed material to vary throughout the river system. The ISEDEQI equation numbers are the same as ISEDEQG above. The number of size fraction intervals must be identical for all sediment groups (see comment 6).

Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ISEDISPLAY	i	1 - NNOD	Grid element (channel or floodplain) for which the sediment transport capacity for all the sediment transport equations will be displayed by output interval TOUT in the SEDTRAN.OUT file. Please note that only one equation is used in the actual sediment routing calculations, but the results of all equations are displayed in SEDTRAN.OUT.
ISEDGRID(N)	i	1 - NNOD	Grid element that will be a sediment supply node (channel or floodplain) with a sediment rating curve.
ISEDGROUP(N)	i	1- NNOD	The sediment group ID for each set of size fraction data. See comment 5.
ISEDSIZEFRAC	s	0 or 1	ISEDSIZEFRAC = 1, the sediment routing will be performed by size fraction. Requires data input from Lines 3 and 4 and Line 9 if a sediment supply is also input. ISEDSIZEFRAC = 0, the default bed thickness is 10 ft (3 m) (see comment 1).
ISEDSUPPLY	s	0 or 1	ISEDSUPPLY = 1 if a sediment rating curve will be used to define the sediment supply to a channel reach or floodplain area.
ISEDUM	i	1 - NNOD	Grid element number with the sediment size fraction group
JDEBNOD	i	1 - NNOD	Grid element with the debris basin.
SCOURDEP	i	0 - 100 0 - 30	Maximum allowable scour depth (ft or m) for all floodplain elements.

Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
SEDCHAR	c	M C Z P D E R S N G	Character line identifier: Set SEDCHAR = 'M' to input mudflow parameters in Line 1. Set SEDCHAR = 'C' to input sediment routing parameters in Line 2. Set SEDCHAR = 'Z' to input sediment routing by size fraction control parameters in Line 3. Set SEDCHAR = 'P' to input sediment routing by size fraction sediment distribution variables in Line 4. Set SEDCHAR = 'D' to input debris basin parameters in Line 5. Set SEDCHAR = 'E' to input the sediment scour limitation parameter in Line 6. Set SEDCHAR = 'R' to input rigid bed grid elements in Line 7. Set SEDCHAR = 'S' to input sediment supply rating curves in Line 8. Set SEDCHAR = 'N' to input sediment supply rating curve size fraction distribution in Line 9. Set SEDCHAR = "G" to input a sediment group.
SEDIAM	r	0 - ∞	Representative sediment diameter (mm) for sediment routing by size fraction. The sediment diameter corresponds to a given size fraction percent finer and usually is a pan sieve size.
SEDPERCENT	r	0. - 1.	Sediment size distribution percentage (expressed as a decimal). The percentage represents the percent of the total sediment that is finer than the representative size diameter. For example, 45.6 percent of the sediment is finer than the 1 mm sediment size fraction. The last entry should be 1.00 (100% of the sediment is smaller than the corresponding sediment diameter).
SGRAD	r	1.0 - 10.	Sediment gradation coefficient (non-dimensional) for the sediment transport routine.
SGSM	r	2.5 - 2.8	Specific gravity of the sediment in the mudflow.
SGST	r	2.6 - 2.8	Sediment specific gravity in the sediment transport component.
SSEDIAM	r	0 - ∞	Representative sediment supply diameter (mm) for sediment routing by size fraction. See SEDIAM parameter above.
SSEDPERCENT	r	0 - 1	Sediment supply size distribution percentage (expressed as a decimal). The percentage represents the percent of the total sediment that is finer than the representative size diameter. See SEDPERCENT parameter above.

Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
VA	r	0 - ∞	Coefficient in the viscosity versus sediment concentration by volume relationship. The relationship is based on a viscosity given in poises (dynes-s/cm^2) for either the English or Metric system (see comment 4).
VB	r	0 - ∞	Exponent in the viscosity versus sediment concentration by volume relationship.
XKX	r	24 - 50,000	The laminar flow resistance parameter for overland flow. This value should range from 24 to 50,000 (see Table 8 in the FLO-2D Reference manual). It is suggested that a value of 2,480 initially be used for mudflows. If a value of XKX is entered, it will be used by the model. If XKX = 0, then XKX is computed by the following formulas where FPN is the floodplain grid element Manning's n-value: FPN < 0.01 XKX = 24 0.01 < FPN < 0.25 XKX = 1,460,865.81* $(FPN)^{2.381}$ 0.25 < FPN XKX = 2,480
YSA	r	0 - ∞	Coefficient of the yield stress versus sediment concentration by volume relationship. The relationship is based on a yield stress given in dynes/cm2 for either the English or Metric system.
YSB	r	0 - ∞	Exponent of yield stress versus sediment concentration by volume relationship.

Instructional Comments for the SED.DAT File

1. Armoring is simulated for bed material sizes with a D_{90} > 16 mm. If D_{90} > 16 mm, then an armor exchange layer with a thickness (3 x D_{90}) is established. Initially the exchange layer has the same sediment size distribution as prescribed for the bed. The volume and size distribution of each sediment size fraction in the exchange layer is tracked on a timestep basis independent of the remaining bed material size. A potential armor sediment size D_{84} is predicted for the prescribed bed material size (see the armor discussion in chapter 4 of the FLO-2D Reference Manual for references). If the computed D_{84} grain size equals or exceeds the predicted D_{84} armor size then an armor layer is assumed that will protect the smaller size sediment in bed from scour.

2. While the bed thickness can be used to limit scour in the channel, it is suggested that a reasonable bed thickness be initially specified to determine if the channel computes an unreasonable scour depth.

3. To select an appropriate sediment routing equation refer to chapter 4 of the FLO-2d Reference Manual. If you are uncertain as to which equation may be best suited to your project, Zeller and Fullerton or Yang's equation will predict a moderate sediment transport capacity for a wide range of field conditions.

4. Mudflow simulation is dependent on the appropriate selection of viscosity and yield stress parameters. Please review the mudflow discussion in Chapter 4 of the Ref Manual to determine an appropriate viscosity and yield stress relationship as function of sediment concentration.

5. The floodplain spatially variable sediment size fraction is assigned by sediment groups (Lines 3, 4 and 10). Line 10 (G) relates the cell number to the sediment group. Spatial variation can be assigned to the channel by segments using the ISEDN parameter in the CHAN.DAT file. ISEDN is used to identify the sediment group for each segment. If there are two sediment groups as shown in the above example data file, there could be one floodplain sediment size distribution and one channel size distribution or there could be two channel segment size distributions by using the first sediment group to represent one of the channel segments as specified by the ISEDN variable in CHAN.DAT.

6. It is important to note that each sediment group will have the identical size fraction delineation. The SEDIAM variable will be the same for all the groups (i.e. the number of Line 4s in all groups will be the same). If one group is missing a specific size fraction, then the sediment percentage for that group (SEDPERCENT variable) will either be the same as the previous value or only slightly different (see the above example data file).

FILE: LEVEE.DAT

LEVEE AND FAILURE DATA

LEVEE.DAT File Variables

0.00 0	Line 1: **RAISELEV ILEVFAIL**
L 1891	Line 2: **LEVCHAR = 'L' LGRIDNO(L)** *L = number of levee grid elements*
D 4 5029.00	Line 3: **LEVCHAR = 'D' LDIR(L,J) LEVCREST(L,J)** *L = number of levee grid elements J = number of levee directions in grid element*
F 1891	Line 4: **LEVCHAR = 'F' LFAILGRID(LF)** *LF = number of failure grid elements*
W 4 5019.5 27.0 10 1 2 0.5	Line 5: **LEVCHAR = 'W' LFAILDIR(LF,LD) FAILEVEL(LF,LD) FAILTIME(LF,LD) LEVBASE(LF,LD) FAILWIDTHMAX(LF,LD) FAILRATE(LF,LD) FAILWIDRATE(LF,LD)** *LD = number of fail directions and LF = number of failure grid elements*
C FS3 0.5	Line 6: **LEVCHAR = 'C' GFRAGCHAR GFRAGPROB**
P 3450 FS1 0.5	Line 7: **LEVCHAR = 'P' LEVFRAGRID(LP) LEVFRAGCHAR (LP) LEVFRAGPROB(LP)** *LP = number levee grid elements with fragility curve assignments*

Notes:
If LEVEE = 0 in the CONT.DAT file, omit this file.
Line 2: Repeat this line for each levee grid element.
Line 3: Repeat this line for each levee direction in a grid.
Line 4: Repeat this line for each LEVEEFAILURE grid element.
Line 5: Repeat this line for each grid element failure direction.

FILE: LEVEE.DAT

LEVEE DATA

LEVEE.DAT File Example

```
  0.00   0
L  1891
D    4  5020.00
L  1896
D    6  5020.00
L  1897
D    2  5020.00
D    3  5020.00
D    5  5020.00
D    6  5020.00
L  1921
D    1  5020.00
D    4  5020.00
D    8  5020.00
L  1922
D    8  5020.00
L  1927
D    2  5020.00
D    6  5020.00
L  ...
C  FS3       0.5
P     3450 S1     0.5
P     3558 S1     0.9
P     3559 S2     0.7
P     3669 S3     0.5
P     3670 S4     0.5
P     3782 C1     0.3
P     3783 S1     0.5
P     3815 J2     0.5
P     3897 S1     0.5
P     ...
```

Variable Descriptions for the LEVEE.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
FAILEVEL(LF,LD)	r	0.01 to ∞	The maximum elevation of the prescribed levee failure if different than the levee crest (LEVELEV). Set FAILEVEL = 0 to fail the levee when overtopped.
FAILRATE(LF,LD)	r	.01 - 1000 0.25 - 300	The rate of vertical levee failure (ft/hr or m/hr).
FAILTIME(LF,LD)	r	0 to SIMUL	The duration (hr) that the levee will fail after the FAILEVEL elevation is exceeded by the flow depth. Set FAILTIME = 0 if the levee fails immediately when overtopped.
FAILWIDRATE(LF,LD)	r	.01 - 1000 0.25 - 300	The rate at which the levee breach widens (ft/hr or m/hr). The breach stops increasing if the breach exceeds the grid element width for that direction.
FAILWIDTHMAX(LF,LD)	r	0 - ∞	The maximum breach width (ft or m). The breach can extend into more than one grid element direction if necessary and the failure width can be larger than one grid element. See Comment 3.
GFRAGCHAR	c	Alpha Numeric	Global levee fragility curve ID. One letter (e.g. S) and one number (e.g. 3) and must correspond to a levee fragility curve ID in the BREACH.DAT file.
GFRAGPROB	r	0 - 1	Global levee fragility curve failure probability. This is assigned to all levee grid elements. The levee fragility curves must be assigned in BREACH.DAT.
ILEVFAIL	s	0, 1 or 2	Switch to identify levee failure mode. See comment 6. Set ILEVFAIL = 0 for no levee failure. Set ILEVFAIL = 1 for prescribed level failure rates of breach opening. Set ILEVFAIL = 2 for initiating the levee or dam breach failure routine.
LDIR(L,IM)	i	1 - 8	Flow direction (of the 8 possible overland flow directions) that will be cutoff by a levee in a given grid element. The possible flow directions are: 1 = north 5 = northeast 2 = east 6 = southeast 3 = south 7 = southwest 4 = west 8 = northwest

Variable Descriptions for the LEVEE.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
LEVBASE(LF,LD)	r	0.01 to LEVCREST	The elevation of the levee crest after complete failure. Set LEVBASE = 0 if levee failure results in the levee crest being reset to the floodplain elevation in the FPLAIN.DAT file. If levee failure exceeds LEVBASE elevation, the floodplain elevation is set to LEVBASE.
LEVCHAR(L)		L, D, F, W, C or P	Character Identifier for Lines 2 - 7 'L' = Line 2 'D' = Line 3 'F' = Line 4 'W' = Line 5 'C' = Line 6 'P' = Line 7
LEVCREST(L,IM)		.01 - 30,000 .25 - 9,000	The elevation of the levee crest (ft or m) (see comments 4 and 5).
LFAILDIR(LF,LD)	i	1 - 8	The potential failure direction (see comment 3). 1 = north 5 = northeast 2 = east 6 = southeast 3 = south 7 = southwest 4 = west 8 = northwest
LEVFRAGRID(LP)	i	1 - NNOD or -1 to -NNOD	Individual levee grid element with fragility curve assignment. The fragility curves must be assigned in BREACH.DAT
LFAILGRID(LF)	i	1 - NNOD	The floodplain grid element number with a levee that may potentially fail. If the LFAILGRID variable is assigned as a negative value, then the levee failure criteria assigned to all levee elements. See comment 1.
LGRIDNO(L)	i	1 - NNOD	The grid element number containing the levee segment . See comment 2.
RAISELEV	r	0 - 100 0 - 30	Incremental height (ft or m) that all the levee grid element crest elevations are raised.

Instructional Comments for the LEVEE.DAT File

1. The general operating rules for the prescribed levee failure are as follows:
   a. To make the levee fails when overtopped by the flow, set FAILELEV and FAILTIME = 0.
   b. If the levee is to fail a at specified elevation, set FAILELEV equal to the failure elevation.
   c. If the levee is to fail a at specified level below the top of the levee, set FAILELEV to a value lower than 10ft. This will cause the levee to fail at an elevation equal to LEVCHREST - FAILELEV.
   d. If the levee is to fail at a specific level below the top of the levee after being inundated for a specific duration, assign appropriate values to both the FAILEVEL and FAILTIME variables.
   e. If the levee fails to a new base elevation that is different than the floodplain elevation, assign an appropriate elevation to LEVBASE.
   f. After levee failure, flows will occur through a breach width of FAILWIDTHMAX for that flow direction. If the maximum failure width is longer than the grid element side, the breach will extend into adjacent grid elements until the maximum failure width is equaled or the levee ends. For instantaneous failure, the entire grid element side levee will collapse.
   g. Progressive levee failure is simulated by assigning a value to FAILRATE (ft/hr). This computes the new levee crest elevation as failure proceeds. FAILRATE is a vertical rate of decrease in the levee breach elevation.
   h. If prescribed failure levee grid element is negative, the failure data for that element is assumed to be global and applies to all the levee elements and blocked flow directions. In this case, the failure data needs only to be assigned to one element.
2. No multiple channels will be assigned to grid elements with levees. Multiple channels in a levee grid element are eliminated automatically by the model.
3. Each levee grid element can have up to eight failure directions. The initial breach widths are hardwired as 1.0 ft (0.3 m). The user specifies the maximum anticipated breach width with the parameter FAILWIDTHMAX. If the maximum failure width is longer than the grid element side, the breach will extend into adjacent grid elements until the maximum failure width is equaled or the levee ends.
4. Flow over a levee is computed as broadcrested weir flow using a coefficient of 2.85 until the tailwater depth is 85% of the headwater depth. The flow then reverts to overland flow based on the water surface elevations on each side of the levee and the flow depth over the levee.

5. Levee freeboard deficit is reported in the output file LEVEEDEFIC.OUT. Five levels of freeboard deficit are listed in the file as follows:

Level 0	> 3 ft
1	2 ft < freeboard < 3 ft
2	1 ft < freeboard < 2 ft
3	freeboard < 1 ft
4	levee overtopped

6. There two options for specifying levee or dam breach failure. Set ILEVFAIL = 1 to assess the breach failure with prescribed rates of breach opening vertically and horizontally. Set ILEVFAIL = 2 to allow the model to simulate the breach geotechnical failure using a modified NWS BREACH model routine.

7. A second document on levee failure can be found in the Handouts folder. C:\users\public\public documents\FLO-2D PRO Documentation\flo_help\Handouts\Levee failure guidelines.pdf.

FILE: FPXSEC.DAT

FLOODPLAIN CROSS SECTION DATA

FPXSEC.DAT File Variables

P 0	Line 1:	**FPXSECHAR = 'P' NXPRT**
X 3 11 284 ...	Line 2:	**FPXSECHAR = 'X' IFLO(N) NNXSEC(N) NODX(N,J)** *J = 1, NNXSEC*

Notes:
Line 2: Repeat this line for each cross section.

FPXSEC.DAT File Example

```
P 0
X 3 11 284 285 286 287 288 289 290 291 292 293 294
X 3 14 808 809 810 811 812 813 814 815 816 817 818 819 820 821
X 3 15 1097 1098 1099 1100 1101 1102 1103 1104 1105 1106 1107 1108 1109 1110 1111
X 3 10 1365 1366 1367 1368 1369 1370 1371 1372 1373 1374
X 3 26 1857 1858 1859 1860 1861 1862 1863 1864 1865 1866 1867 1868 1869 1870 1871
X 3 28 2491 2492 2493 2494 2495 2496 2497 2498 2499 2500 2501 2502 2503 2504 2505
X 3 12 4224 4225 4226 4227 4228 4229 4230 4231 4232 4233 4234 4235
X 2 8 7373 7303 7236 7180 7124 7068 7012 6956
X 2 5 8233 8135 7941 7845 7749
X 3 6 9000 9001 9002 9003 9004 9005
X 3 ...
```

Variable Descriptions for the FPXSEC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
FPXSECHAR	c	P or X	Character line identifier for Lines 2 and 3, 'P' = Line 2 'X' = Line 3
IFLO(N)	i	1 - 8	Defines the general direction that the flow is expected to cross the floodplain cross section. See comment 1. IFLO is set to one of the following: 1 flow to the north 5 flow to the northeast 2 flow to the east 6 flow to the southeast 3 flow to the south 7 flow to the southwest 4 flow to the west 8 flow to the northwest If the output is desired from only one direction (i.e. without the discharge components from the other component flow directions), set IFLO as negative. IFLO is set to the following: -1 flow from south only -5 flow from southwest only -2 flow from west only -6 flow from northwest only -3 flow from north only -7 flow from northeast only -4 flow from east only -8 flow from southeast only
NODX(N,J)	i	1 - NNOD	Array of grid elements that constitute a given floodplain cross section (see comment 2 and 3).
NNXSEC(N)	i	1 - 1,000	Number of floodplain elements in a given cross section. The selected cross section grid elements do not have to extend across the entire grid system. Only one grid element is necessary to constitute a floodplain cross section. The cross section can include a channel element. If one of the floodplain cross section grid elements is a channel element, the cross section discharge hydrograph reported in HYCROSS will include the channel element discharge.
NXPRT	s	0 or 1	If NXPRT = 1, the cross section summary information including cross section discharge, average cross section velocity, width and depth will be reported in the BASE.OUT file.

Instructional Comments for the FPXSEC.DAT File

1. The floodplain grid elements can be combined to define a cross section across a floodplain or alluvial fan. Each floodplain cross section is assessed flow discharge in only one flow direction given by IFLO. This direction includes the flow contribution from the two contiguous directions. The cross section routine can be used to isolate the results for a single element. The flow directions and associated discharge components are as follows:

CROSS SECTION FLOW DIRECTION DATA	
Selected Cross Section Flow Direction	**Flow Direction Components added to the Cross Section Discharge**
north = 1	northeast 5 and northwest 8
east = 2	northeast 5 and southeast 6
south = 3	southeast 6 and southwest 7
west = 4	southwest 7 and northwest 8
northeast = 5	north 1 and east 2
southeast = 6	east 2 and south 3
southwest = 7	south 3 and west 4
northwest = 8	west 4 and north 1

For the diagonal flow directions (5 thru 8), the discharge for the grid element between the two diagonal corners will be added to the cross section total discharge for the selected flow direction.

2. If a grid element is listed in more than one cross section, then the individual grid element hydrographs in the output file CROSSQ.OUT will not be correct. Output data in the CROSSMAX.OUT and HYCROSS.OUT files, however, will be correct.

3. The floodplain cross section grid elements can be selected graphically with the GDS program.

FILE: BREACH.DAT

DAM AND LEVEE BREACH DATA

BREACH.DAT File Variables

Line 1: **IBR = 'B1' IBREACHSEDEQN GBRATIO GWEIRCOEF GBREACHTIME**

B1 4 2.0 2.95 0.50

Line 2: **IBR = 'G1' GZU GZD GZC GCRESTWIDTH GCRESTLENGTH GBRBOTWIDMAX GBRTOPWIDMAX GBRBOTTOMEL**

G1 2.0 2.0 0. 5. 0. 0. 0. 1.5

Line 3: **IBR = 'G2' GD50C GPORC GUWC GCNC GAFRC GCOHC GUNFCC**

G2 0. 0. 0. 0. 0. 0. 0.

Line 4: **IBR = 'G3' GD50S GPORS GUWS GCNS GAFRS GCOHS GUNFCS**

G3 0.25 0.40 100. 0.06 30. 65. 0.

Line 5: **IBR = 'G4' GGRASSLENGTH GGRASSCOND GGRASSVMAXP GSEDCONMAX D50DF GUNFCDF**

G4 4. 1. 4. 0. 0. 0.

Line 6: **IBR = 'B2' IBREACHGRID IBREACHDIR**

B2 4015 7

Line 7: **IBR = 'D1' ZU ZD ZC CRESTWIDTH CRESTLENGTH BRBOTWIDMAX BRTOPWIDMAX BRBOTTOMEL WEIRCOEF**

D1 2.0 2.0 0. 8. 0. 0. 0. 83.25 3.05

Line 8: **IBR = 'D2' D50C PORC UWC CNC AFRC COHC UNFCC**

D2 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.

Line 9: **IBR = 'D3' D50S PORS UWS CNS AFRS COHS UNFCS**

D3 0.25 0.40 100. 0.10 25. 100. 0.

Line 10: **IBR = 'D4' BRATIO GRASSLENGTH GRASSCOND GRASSVMAXP SEDCONMAX D50DF UNFCDF BREACHTIME**

D4 0. 0. 0. 0. 0. 0. 0.

Line 11: **IBR = 'F' FRAGCHAR(I) PRFAIL(I,J) PRDEPTH(I,J);**

I = number of levee fragility curves and J = number of points in each fragility curve

F S1 0.03 6.0

FILE: BREACH.DAT

DAM AND LEVEE BREACH DATA

BREACH.DAT File Example

Notes:

Line 1: Required for a sediment erosion breach

Lines 2 - 5: Global data required to locate a breach. Not required for a prescribed breach location.

Lines 6 - 10: Optional data for prescribed breach location. Repeat these lines for each specified breach grid element.

Line 10: Repeat this line for each fragility curve listing

```
B1  4.0  2.0  2.95   0.50
G1 2.0   2.0   0.   5.   0.   0.   0.   3.   3.05
G2 0.    0.    0.   0.   0.   0.   0.
G3 0.25  0.40  100.   0.06  30.   100.   0.
G4 1.    0.    0.   0.   0.   0.   0.
B2  4015  7
D1  2.0   2.0   0.   8.   0.   0.   0.   83.25   3.05
D2 0.    0.    0.   0.   0.   0.   0.   0.   0.   0.
D3  0.25   0.40   100.   0.10  25.   100.   0.
D4  2.    0.    0.   0.   0.   0.   0.   0.
F S1  0.03  6.0
F S1  0.15  3.5
F S1  0.50  2.5
F S1  0.85  1.0
F S1  0.95  0.0
F S2  0.03  9.0
F S2  0.15  5.5
F S2  0.50  4.0
F S2  0.85  2.0
F S2  0.98  0.0
F S3  0.03 12.0
F S3  ...
```

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
AFRC	r	0 - 50	Angle (degrees) of internal friction of the core material for failure of a specific grid element flow direction. Set AFRC = 0.0 for no core.
AFRS	r	0 - 50	Angle (degrees) of internal friction of the shell material for failure of a specific grid element flow direction.
BRATIO	r	1 - 5	Ratio of the initial breach width to flow depth. See comments 2 and 3.
BRBOTTOMEL	r	0 - ∞	Initial breach or pipe bottom elevation (ft or m). See comments 5 and 6.
BRBOTWIDMAX	r	0 - ∞	Maximum allowable breach bottom width (ft or m) as constrained by the valley cross section. Set BRBOTWIDWAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side).
BREACHTIME	r	0 - ∞	The duration (hrs) that the levee erosion will initiate after the water surface exceeds the specified pipe elevation BRBOTTOMEL. set BREACHTIME = 0 if the level erosion begins immediately when pipe elevation is exceeded.
BRTOPWIDMAX	r	0 - ∞	Maximum allowable breach top width (ft or m) as constrained by the valley cross section. Set BRTOPWIDMAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side).
COHC	r	0 - 750 0 - 30,000	Cohesive strength (lb/ft^2 or N/m^2) of the levee or dam core material. If there is no core, COHC = 0.
COHS	r	0 - 750 0 - 30,000	Cohesive strength (lb/ft^2 or N/m^2) of the levee or dam shell material. If there is no core, COHS = 0.
CNC	r	0.02 - 0.25	Manning's n-value of the levee or dam core material. If CNC = 0., Manning's n-value for the core material will computed from Strickler's equation. If CNC > 1., the n-value will be computed from a Moody diagram (Darcy f vs. D_{50}). Set CNC = 0.0 for no core material.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
CNS	r	0.02 - 0.25	Manning's n-value of the levee or dam shell material. See comment 4. If CNS = 0., Manning's n-value for the shell material will computed from Strickler's equation. If CNS > 1., the n-value will be computed from a Moody diagram (Darcy f vs. D_{50}).
CRESTLENGTH	r	0 - ∞	Length of the crest of the levee or dam (ft or m). If CRESTLENGTH = 0., the crest length will default to the grid element octagon side. If crest length is greater than the grid element octagon side, it will be reset to the octagon side length.
CRESTWIDTH	r	0 - ∞	Crest width of the levee or dam (ft or m). The crest width can be zero.
D50C	r	0.0625 - 2	Mean sediment size (D_{50} in mm) of the levee or dam core material.
D50S	r	0.25 - 10	Mean sediment size (D_{50} in mm) of the levee or dam shell material.
D50DF	r	1.0 - 100	Mean sediment size (D_{50} in mm) of the top one foot (0.3 m) of the downstream face (riprap material). If D50DF = 0.0, then D50DF = D50S.
FRAGCHAR	c	S1, S2 ...	Fragility curve ID. One letter and a number. For example: S1 is fragility curve 1 for the Sacramento River (see comment 7).
GAFRC	r	0 - 50	Global angle (degrees) of internal friction of the core material for the entire levee or dam. Set AFRC = 0.0 for no core.
GAFRS	r	0 - 50	Global angle (degrees) of internal friction of the shell material for the entire levee or dam.
GBRATIO	r	1 - 5	Global ratio of the initial breach width to flow depth for an unspecified failure location.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
GBRBOTTOMEL	r	0 - ∞	Initial global breach or pipe bottom elevation (ft or m) for an unspecified failure location. If the model will locate the failure grid element instead of user specified failure location, then set GBRBOTTOMEL = distance below the dam or levee crest elevation (ft or m). In general, GBRBOTTOMEL be less than 10 ft (3 m) (see comments 1 and 6).
GBRBOTWIDMAX	r	0 - ∞	Maximum allowable global breach bottom width (ft or m) as constrained by the valley cross section for an unspecified failure location. Set GBRBOTWIDWAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side).
GBREACHTIME	r	0 - ∞	The duration (hrs) that the levee erosion will initiate after the water surface exceeds the specified pipe elevation BRBOTTOMEL. set BREACHTIME = 0 if the level erosion begins immediately when pipe elevation is exceeded.
GBRTOPWIDMAX	r	0 - ∞	Maximum allowable global breach top width (ft or m) as constrained by the valley cross section for an unspecified failure location. Set GBRTOPWIDMAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side).
GCOHC	r	0 - 750 0 - 30,000	Global cohesive strength (lb/ft^2 or N/m^2) of the levee or dam core material for an unspecified failure location. If there is no core, GCOHC = 0.
GCOHS	r	0 - 750 0 - 30,000	Global cohesive strength (lb/ft^2 or N/m^2) of the levee or dam shell material for an unspecified failure location. If there is no core, GCOHS = 0.
GCNC	r	0.03 - 0.1	Global Manning's n-value of the levee or dam core material for an unspecified failure location. See comment 4. If GCNC = 0.0, Manning's n-value for the core material will computed from Strickler's equation. Results in a very low n-value. Not recommended. If GCNC > 1., the n-value will be computed from a Moody diagram (Darcy f vs. D_{50}). Set GCNC = 0.0 for no core material.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
GCNS	r	0.03 - 0.1	Global Manning's n-value of the levee or dam shell material for an unspecified failure location. See comment 4. If GCNS = 0., Manning's n-value for the shell material will computed from Strickler's equation. Not recommended. If GCNS > 1., the n-value will be computed from a Moody diagram (Darcy f vs. D_{50}).
GCRESTLENGTH	r	0 - ∞	Global crest length of the levee or dam (ft or m) for an unspecified failure location. If GCRESTLENGTH = 0.0, the crest length will default to the grid element octagon side. If crest length is greater than the grid element octagon side, it will be reset to the octagon side length.
GCRESTWIDTH	r	0 - ∞	Global crest width of the levee or dam (ft or m) for an unspecified failure location. The crest width can be zero.
GD50C	r	0.0625 - 2	Mean sediment size (D_{50} in mm) of the levee or dam core material.
GD50S	r	0.25 - 10	Mean sediment size (D_{50} in mm) of the levee or dam shell material.
GD50DF	r	1 - 100	Mean sediment size (D_{50} in mm) of the top one foot (0.3 m) of the downstream face (riprap material). If GD50DF = 0.0, then GD50DF = GD50S.
GGRASSCOND	r	0 - 1	Global condition of the grass on the downstream face of the levee or dam for an unspecified failure location. 0.0 for a poor stand or no grass;1.0 for a good stand of grass.
GGRASSLENGTH	r	0 - 10	Global average length of grass (inches or mm) on downstream face for an unspecified failure location. Set GGRASSLENGTH = 0.0 for no grass on downstream face.
GGRASSVMAXP	r	3 - 6 1 - 2	Global maximum permissible velocity (fps or mps) for a grass-lined downstream face before the grass is eroded for an unspecified failure location. Range: 3 to 6 fps (1 to 2 mps). If no grass, set GGRASSVMAXP = 0.0.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
GPORC	r	0.35 - 0.45	Global porosity of the levee or dam core material for an unspecified failure location. Typical range: 0.35 to 0.45. Set GPORC = 0.0 for no core material.
GPORS	r	0.35 - 0.45	Global porosity of the levee or dam shell material for an unspecified failure location. Typical range: 0.35 to 0.45.
GRASSCOND	r	0 - 1	Condition of the grass on the downstream face of the levee or dam for a prescribed failure location. 0.0 for a poor stand or no grass; 1.0 for a good stand of grass.
GRASSLENGTH	r	0 - 1 0 - 25	Average length of grass (inches or mm) on downstream face for a prescribed failure location. Set GRASSLENGTH = 0.0 for no grass on downstream face.
GRASSVMAXP	r	3 - 6 1 - 2	Maximum permissible velocity (fps or mps) for a grass-lined downstream face before the grass is eroded for a prescribed failure location. Range: 3 to 6 fps (1 to 2 mps). If no grass, set GRASSVMAXP = 0.0.
GSEDCONMAX	r	0.2 - 0.55	Global maximum sediment concentration by volume in the breach discharge for an unspecified failure location. Typical range = 0.2 to 0.55. If GSEDCONMAX = 0.0, a default value of 0.5 is used.
GUNFCC	r	1 - 20	Global sediment gradient, ratio of D_{90} to D_{30} of the levee or dam core material for an unspecified failure location. If there is no core material, set GUNFCC = 0.0. If the there is core material and GUNFCC = 0.0, it is reset to 10.0.
GUNFCDF	r	1 - 20	Global sediment gradient, ratio of D_{90} to D_{30} of the downstream face upper one foot of material (riprap) for an unspecified failure location. If GUNFCDF = 0.0: GUNDFCDF = GUNFCS when GD50DF = 0.0 And GUNDFCDF = 3.0 when GD50DF > 0.0.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
GUNFCS	r	1 - 20	Global sediment gradient, ratio of D_{90} to D_{30} of the levee or dam shell material for an unspecified failure location. If GUNFCS = 0.0, the default value is 10.0.
GUWC	r	85 - 120 13,500 - 19,000	Global unit weight (lb/ft^3 or N/m^3) of the levee or dam core material for an unspecified failure location. Set GUWC = 0.0 if there no core.
GUWS	r	85 - 120 13,500 - 19,000	Global unit weight (lb/ft^3 or N/m^3) of the levee or dam shell material for an unspecified failure location.
GWEIRCOEF	r	2.85 - 3.05	Global weir coefficient for piping or breach channel weir for an unspecified failure location. Typical range: 2.85 – 3.05.
GZC	r	0.1 - 10	Global average slope of the upstream and downstream face of the levee or dam core material for an unspecified failure location. GZC is expressed as a ratio of the GZC (horizontal:1 (vertical). For example: GZC = 2.0 represents 2.0 horizontal to 1.0 vertical. If there is no core set GZC = 0.0
GZD	r	0.1 - 10	Global slope of the downstream face of the levee or dam for an unspecified failure location. GZD is expressed as a ratio of the GZD (horizontal : vertical). For example: GZD = 2.0 represents 2.0 horizontal to 1.0 vertical.
GZU	r	0.1 - 10	Global slope of the upstream face of the levee or dam for an unspecified failure location. GZU is expressed as a ratio of the GZU (horizontal : vertical). For example: GZU = 2.0 represents 2.0 horizontal to 1.0 vertical.
IBR	c	B, D1, D2, D3, D4, G1, G2, G3, G4 or F	Character line identifier 'G1-G4' = global data 'B' = grid element and direction data; and 'D1-D4' = individual prescribed grid element breach data. 'F' = fragility curve data

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
IBREACHDIR	i	1 - 8	Direction of the specified breach failure in a given grid element. The possible flow directions are: 1 = north 5 = northeast 2 = east 6 = southeast 3 = south 7 = southwest 4 = west 8 = northwest
IBREACHGRID	i	1 - NNOD	Grid element of the specified breach failure location. See comment 8.
IBREACHSEDEQN	i	1 - 11	Sediment transport equation that is used to compute the breach erosion. Out of eleven transport equations in FLO-2D only Tofaletti and MPM-Woo are not available. See the list of sediment tranport equation numbers in SED.DAT.
PORC	r	0.35 - 0.45	Porosity of the levee or dam core material for a prescribed grid element failure location. Set GPORC = 0.0 for no core material.
PORS	r	0.35 - 0.45	Porosity of the levee or dam shell material for an prescribed grid element failure location.
PRDEPTH	r	0.0 - Levee Crest Height	Point of failure on the levee as defined by the distance or height below the levee crest (likely failure point according to the Corps of Engineers definition). Assigned with a corresponding fragility curve failure probability PRFAIL.
PRFAIL	r	0.0 - 1.0	Levee fragility curve point of failure probability. Range: 0.0 to 1.0 where 80% indicates a higher probability of levee failure corresponding to a higher elevation on the levee (see the levee fragility curve discussion in the FLO-2D Reference Manual). A low value of 10% would indicate a weak levee corresponding to a levee piping failure initial when the water surface elevation is close to the toe of the levee.
SEDCONMAX	r	0.20 - 0.55	Maximum sediment concentration by volume in the breach discharge for a prescribed grid element failure location. Typical range = 0.2 to 0.55. If SEDCONMAX = 0.0, a default value of 0.5 is used.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
UNFCC	r	1 - 20	Sediment gradient, ratio of D_{90} to D_{30} of the levee or dam core material for a prescribed grid element failure location. If there is no core material, set UNFCC = 0.0. If the there is core material and UNFCC = 0.0, it is reset to 10.0.
UNFCDF	r	1 - 20	Sediment gradient, ratio of D_{90} to D_{30} of the downstream face upper one foot of material (riprap) for a prescribed grid element failure location. If UNFCDF = 0.0 : UNDFCDF = UNFCS when D50DF = 0.0 And UNDFCDF = 3.0 when D50DF > 0.0.
UNFCS	r	1 - 20	Sediment gradient, ratio of D90 to D30 of the levee or dam shell material for a prescribed grid element failure location. If UNFCS = 0.0, the default value is 10.0.
UWC	r	85 -120 13,500 - 19,000	Unit weight (lb/ft^3 or N/m^3) of the levee or dam core material for a prescribed grid element failure location. Set UWC = 0.0 if there no core.
UWS	r	85 - 120 13,500 - 19,000	Unit weight (lb/ft^3 or N/m^3) of the levee or dam shell material for a prescribed grid element failure location.
WEIRCOEF	r	2.85 - 3.05	Weir coefficient for piping or breach channel weir for a prescribed grid element failure location. Typical range: 2.85 – 3.05.
ZC	r	0.1 - 10	Average slope of the upstream and downstream face of the levee or dam core material for a prescribed failure location. ZC is expressed as a ratio of the ZC (horizontal : vertical). For example: ZC = 2.0 represents 2.0 horizontal to 1.0 vertical. If there is no core set ZC = 0.
ZD	r	0.1 - 10	Slope of the downstream face of the levee or dam for a prescribed grid element failure location. ZD is expressed as a ratio of the ZD (horizontal : vertical). For example: ZD = 2.0 represents 2.0 horizontal to 1.0 vertical.

Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
ZU	r	0.1 - 10	Slope of the upstream face of the levee or dam for a prescribed grid element failure location. ZU is expressed as a ratio of the ZU (horizontal : vertical). For example: ZU = 2.0 represents 2.0 horizontal to 1.0 vertical.

Instructional Comments for the BREACH.DAT File

1. You have the choice of identifying a breach location or letting the model determine the breach location by water surface elevation. If you assign the breach location, it is necessary to provide all the lines of data in the BREACH.DAT file. If you allow the model to locate all the potential breach locations by the water surface elevation, then it is only necessary assign the global parameters in lines 1-4. In addition, set GBRBOTTOMEL = vertical distance (ft or m) below the levee or dam crest elevation. If the water surface elevation exceeds the elevation of the crest – GBRBOTTOMEL, then a levee piping failure will be initiated. You can prescribe one or more breach locations and permit the model to determine any other potential breach locations by setting GBBOTTOMEL to value less than about 10 ft (3 m) below the crest elevation.

2. Initial breach width to depth ratio (BRATIO) – if the assigned breach width to depth ration is 0., then BRATIO = 2.

3. The initial piping width and depth is assumed to be 0.5 ft (0.15 m).

4. The minimum and maximum Manning's n-value permitted for the breach flow resistance are 0.02 and 0.25, respectively.

5. The downstream pipe outlet at the toe of the dam or levee is the grid element floodplain elevation plus 1 ft (0.3 m).

6. Breach discharge is computed if the upstream water surface elevation exceeds the upstream breach pipe or channel bottom elevation plus the tolerance value (TOL ~ 0.1 ft or 0.3 m).

7. The levee fragility curve ID is only one letter (e.g. S) and a number (e.g. 3) and the data line begins with ID character 'F'. The levee fragility curve assignment to the levee grid element is assigned in the LEVEE.DAT file.

8. If Line 2 begins with G1, a global breach simulation is initiated to locate the potential breach based on the water surface elevation. If Line 2 has a B2 identifier, the a prescribed breach location will be simulated as defined by the breach element and direction in Line B2 (global breach data is not required)

FILE: FPFROUDE.DAT

FLOODPLAIN LIMITING FROUDE NUMBERS

FPFROUDE.DAT File Variables

Line 1: **IFR = 'F' I FPFROUDE (I = 1, NNOD)**

F 1 0.65

FPFROUDE.DAT File Example

```
F   1    0.65
F   2    0.88
F   3    0.90
F   43   0.90
F   54   0.90
F   56   1.05
F   107  0.90
F   108  0.90
```

Variable Descriptions for the FPFROUDE.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
IFR	i	1 - NNOD	Grid element number (I) of the floodplain grid system.
Ifr	c	F	Character Line Identifier = 'F'
FPFROUDE	r	0.1 - 2	Floodplain limiting Froude number.

FILE: SWMMFLO.DAT
STORM DRAIN DATA FILE

SWMMFLO.DAT File Variables

Line 1: **SWMMCHAR = 'D' SWMM_JT(I) INTYPE(I) SWMMlength(I) SWMMwidth(I) SWMMheight(I) SWMMcoeff(I) FLAPGATE(I)**

D 14292 2 13 1 0.4167 2.3 0

Line 2: **SWMMCHAR = 'N' DEPTHSWMMRT(J,K) QSWMMRT(J,K)**

N 0.0 0.0

I = number of storm drain inlet nodes.
J= number of storm drain inlets with a rating table.
K= number of rating table pairs.

Notes:
Line 1: IF INTYPE(I) = 4, add line 2 for the rating table.

SWMMFLO.DAT File Example

```
D 14292 2 13 1    0.4167 2.3 0
D 14481 2 13 1    0.4167 2.3 0
D 13785 2 20 1    0.4167 2.3 0
D 14156 4 0. 0.   0.0    0.0 0
N 0.0 0.0
N 1.0 2.1
N 2.0 5.8
N 5.0 25.1
D 14156 3 11 7    0.5    3.0 0
D 14156 3 86.5 10.75 0.5    3.0 0
```

Variable Descriptions for the SWMMFLO.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

VARIABLE	FMT	RANGE	DESCRIPTION
DEPTHSWMMRT	r	0 - ∞	Flow depth (ft or m) above the storm drain inlet grate.
DEPTHSWMMRT(J,K)	r	0 - ∞	Array of flow depths for the discharge rating table pairs.
FLAPGATE(I)	i	0 or 1	Flapgate switch (0 no flap gate, 1 flapgate). A flap gate only allows flow out of the outflow node. No inflow. See comment 2.
INTYPE(I)	i	1 - 4	Type of storm drain inlets. See comment 1.
QSWMMRT	r	0 - ∞	Discharge (cfs or cms) for the storm inlet rating table.
QSWMMRT(J,K)	r	0 - ∞	Array of discharge values for the storm drain inlet rating table.
SWMMCHAR	c	D, N	Character line identifier for the SWMM model inlets.
SWMMcoeff(I)	r	2.50 - 3.50	Array of storm drain inlet weir discharge coefficients.
SWMMheight(I)	r	0.0 - 2.0	Array of storm drain curb opening heights (typically less than 1 ft). For type 3 grate (gutter) inlets, SWMMheight = grate sag height.
SWMM_JT(I)	i	1 - NNOD	Array of grid elements with storm drain inlets.
SWMMlength(I)	r	0.01 - ∞	Array of storm drain inlet curb opening lengths along the curb. For type 3 grate (gutter) inlets, SWMMlength = grate wetter perimeter.
SWMMwidth(I)	r	0 - ∞	Array of storm drain inlet curb opening widths. For type 3 grate (gutter) inlets, SWMMwidth = grate opening area.

Instructional Comments for the SWMMFLO.DAT File

1. The SWMMFLO.DAT file is automatically generated by GDS using the SWMM.inp file data to locate the inlet position and transfer it to the FLO-2D grid system. In the SWMMFLO.DAT file there are three storm drain inlet options:

    1) Curb Opening Inlet at Grade (Type 1)
    2) Curb Opening Inlet Depressed or Sag (Type 2)
    3) Grate (Gutter) Inlet (at Grade or Depressed - Type 3)

The storm drain inlet data requirements are:

Type 1 - Curb Opening Inlet at Grade
Weir coefficient: 2.85 - 3.30 (suggested 3.00 English, 1.6 metric)
Curb Opening Length
Curb Opening Height

Type 2 - Curb Opening Inlet with Sag
Weir coefficient: 2.30 (1.25 metric)
Curb Opening Length
Curb Opening Height
Curb Opening Sag Width

Type 3 - Grate (Gutter) Inlet with/without Sag
Weir coefficient: 2.85 - 3.30 (suggested 3.00 English, 1.6 metric)
Grate Perimeter (not including curb side)
Grate open area
Grate sag height (zero for at grade)

Type 4 - Variable storm drain inlet geometry.
Weir coefficient: Not Required
The storm drain inlet rating table (line n with depth and discharge pairs) is required.

Note: Orifice flow coefficient = 0.67 (hardwired) for all cases.

Table 1. Required Storm Drain Inlet Data in the SWMMFLO.DAT file

Node	Type	Length	Width (sag)	Height	Weir Coeff
X	1	X	0	X	2.85 - 3.20
X	2	X	X	X	2.3
X	3	X wetted perimeter	X clear area	X sag	2.85 - 3.20

where X represents a variable number to be entered by the user.

2. Flapgates on storm drain outlets prohibit the water from flow upstream into the storm drain pipe system. Typically flapgates are used on storm drain outlets into detention basins or channels.

CHAPTER 5

OUTPUT FILES AND OPTIONS

5.1 BASIC GUIDELINES FOR USING OUTPUT FILES

During the flood simulation, the user has two choices for viewing the model flood progression: 1) An output text screen; or 2) A graphical representation of the flood simulation. The screen option is selected with the LGPLOT variable in the CONT. DAT file. The text screen scrolls a list of the simulation time, minimum timestep and the volume conservation data. This screen enables the user to view the simulation runtime speed and the numerical stability of the run. The second screen option is a color graphics display of the grid element maximum flow depth (area of inundation) along with the inflow hydrograph. After the flood simulation is complete, the hydraulic results are presented in output files that contain hydrographs, maximum flow depths and velocities.

The conservation of flood volume (inflow equals outflow plus storage) should be reviewed with each simulation. A summary of the inflow volume, final volumes left on the floodplain (storage) and outflow from the grid system is presented in the SUMMARY.OUT file. If the inflow does not equal the outflow plus final storage within a fraction of one percent, there is problem with the simulation.

Options are available to create output files with either a spatial or temporal format. Output files that have substantial detail involving all the timesteps (temporal output) or all the grid elements (spatial output) can be generated. Some output files are designed for a simple review of specific flow hydraulics. Output data include water surface elevation, flow depth, velocity, discharge, impact pressure, specific energy, sediment concentration and other variables. Overland flow hydraulics may be viewed as individual grid elements or the grid elements can be grouped together

to produce floodplain cross sections. Summary tables listing maximum velocity and flow depths and their times of occurrence appear near the end of the BASE.OUT file. You can specify which type of output you want using the CONT.DAT file. Review the CONT.DAT file description in Chapter 4 for more information about specifying output file formats.

The user is cautioned about specifying the output in too much detail which can result in extremely large output files. The output time interval TOUT and NOPRTFP options in the CONT.DAT file can be assigned to limit the size of the output files. When NOPRTFP is set to 0, all the floodplain output data is written to the BASE.OUT file for each output interval. Setting NOPRTFP = 2 will turn off all of the temporal floodplain hydraulic output data.

> **Hint:**
>
> **Each time a model is run from a specific project folder, all of the *.OUT files are rewritten. To save the model results from a simulation either make copies of the output files or copy the data files to a new project folder and run the model from it.**

The user does not have to specify any output file names. Simply clicking on the run model commands in the GDS menus or double clicking on FLO.EXE in Explorer runs the model and produces the output files. It is important to note that each time FLO-2D is run, it will write over the existing output files and the previous output file data will be lost. To save any output files in anticipation of subsequent FLO-2D runs, the user must move them to another subdirectory.

5.2 OUTPUT FILE DESCRIPTIONS AND OPTIONS

Use WordPad© or any other ASCII editor to view the output files. A brief description of all the output files follows:

BASE.OUT

BASE.OUT is an all enclusive output file. At the beginning of the file, the inflow hydrographs are printed, then the time dependent output data follows. For each specified time output interval, the flow depth, velocity, water surface elevation and discharge for either the channel or the floodplain grid elements can be written. The user will probably have little interest in the floodplain hydraulics for the individual grid elements and this portion of the file should be turned off for most applications (set NOPRTFP = 2). The outflow from the boundary grid elements is listed at the end of the time interval. After the final time output interval, a summary of all the grid elements maximum depths, water surface elevations, velocities and the time of occurrence of the maximum values is printed. Finally, a summary table of the inflow, outflow and storage volumes at the end of the file allows the user to review the conservation of mass and the ultimate disposition of all the water and sediment. For convenience, this conservation table is also written to a separate file named SUMMARY.OUT. There is so much output data in the BASE.OUT file that the user is encouraged to review all the various data output options before starting a flood simulation.

BINARY FILES

The following binary backup files are generated when IBACKUP = 1. These files are used to restart model after termination (either interrupted simulation or end of the simulation).

- CHANBINARY.OUT
- CROSSBINARY.OUT
- FPLAINBINARY.OUT
- HYSTRUCBINARY.OUT
- SEDBINARY.OUT
- STREETBINARY.OUT
- VOLUMEBINARY.OUT
- XSECSEDBINARY.OUT

BREACH.OUT

This file is generated when the breach routine is activated for dams or levees. This output file lists the breach water and sediment discharge, concentration by volume, breach bottom width and elevation as function of the output interval for every grid element and direction in which a levee has breached.

CHANBANKEL.CHK

When the bank elevations prescribed in the CHAN.DAT do not match the corresponding floodplain grid element elevations, the difference in elevation is reported in this file. If the difference between the channel bank elevation and the floodplain grid element elevation exceeds 1 ft (0.3 m), the model reports it in this file. If the 1 ft (0.3 m) limit is exceeded during the simulation, the model assumes that the bank elevation is more accurate and resets the floodplain elevation to the cross section top-of-bank elevation.

CHANMAX.OUT

The maximum discharge and stage for each channel element and the corresponding time of occurrence is written to this file. For mudflows, this file will also list the maximum sediment concentration and the time of occurrence.

CHANNEL.CHK

When the channel cross section width exceeds the grid element width, the cross section extends into neighboring elements. This file provides a list of all channel extensions that are required by a cross section.

CHAN.RGH

CHAN.RGH contains a replica of the CHAN.DAT file with the updated Manning's n-value changes. The maximum and final Manning's n-values changes are reported in the ROUGH.OUT file. To accept the changes to Manning's n-values, CHAN.RGH can then be renamed to CHAN.DAT for the next FLO-2D flood simulation. This automates the spatial adjustment of n-values for channel elements that exceed the limiting Froude number.

CHANSEDSIZE.OUT

The initial and final sediment size distribution by channel element is written to this file.

CHANSTABILTY.OUT

This output file lists the channel grid elements that experienced significant gains or losses of flow volume (0.1 af or 100 m^3). These channel grid elements may have volume conservation stability problems that could be related to surging, poorly matched roughness, slope and cross section geometry or abrupt changes in cross section geometry. When the channel volume conservation for a simulation is not satisfactory, review this output file.

CHANWS.OUT

This output file lists channel grid element, x-coordinate, y-coordinate and maximum channel water surface elevation.

CHNBEDEL.OUT

The channel grid element number and the final channel bed elevation are presented in this file.

CHVOLUME.OUT

The channel volume distribution is listed in this output file including channel inflow, channel outflow, overbank flow, return flow from the floodplain and infiltration and channel storage. Review this file along with the SUMMARY.OUT to determine if the channel flow volume is being conserved.

CONFLUENCE.OUT

This file lists the channel elements that constitute a confluence as defined by having three or more channel elements contiguous to a given channel element.

CROSSMAX.OUT

When the floodplain cross section analysis is requested by creating the FPXSEC.DAT file, the CROSSMAX.OUT is created. This file lists the maximum discharge, maximum flow depth and time of occurrence for each grid element specified in the cross section analysis.

CROSSQ.OUT

This file contains the grid element hydrographs for each of the floodplain elements in the cross section. The time and discharge are listed for each output interval.

DEPTH.OUT and Related Flow Depth Output Files

A series of files are created by FLO-2D in the format: grid element number, x- and y-coordinates, and the maximum flow depth. These files can be viewed with MAXPLOT or MAPPER++ programs or they can be imported to a CADD or GIS program to create maximum flood depth contours. The following output files are created:

CHNBEDEL.OUT - Channel bed elevations
DEPCH.OUT - Maximum channel flow depths
DEPCHFINAL.OUT – Final channel flow depths at the end of the simulation
DEPFP.OUT - Maximum floodplain flow depths
DEPTH.OUT - Maximum combined channel/floodplain flow depths
DEPTHTOL.OUT - Maximum combined channel and floodplain flow depths greater than the TOL value. Values less than the TOL value are set to zero. This file has the following format: x- and y- coordinates, and maximum flow depth. No grid element numbers are included.
FINALDEP.OUT - Final floodplain flow depths.
INTERGWS.OUT - Maximum floodplain water surface elevations based on the original grid element elevation in the TOPO.ORI file. Values less than TOL are set to zero. Only grid element and maximum water surface are listed. No coordinates are included.

DEPTHDUR.OUT and DEPTHDUR2.OUT

DEPTHDUR.OUT contains the floodplain inundation duration data including the grid element number, grid element x- and y-coordinates and duration of inundation in hours. The selected depth of inundation for which the duration (hrs) is computed is listed at the top of the file. DEPTHDUR2.OUT is identical to DEPTHDUR.OUT but for a hardwired depth of 2 ft.

ERROR.CHK

The ERROR.CHK file contains input data error comments when the program terminates prior to the hydraulic computations. Some input data errors that are encountered at execution time are not listed in this file. The backup data files (*.BAC) should be reviewed with this file to determine if the input data is being read properly at runtime. When a simulation terminates immediately after being started, check this file first for data input errors.

FLOODTIME.OUT

This file has contains the following output:

Node X-coord Y-coord Floodwave Arrival Time Flood Time Peak Time Deflood Time Max WS

Each grid element is assigned a specific value of the above parameters at the end of the simulation. The maximum values are tracked during the simulation on a computational timestep basis. The following parameter definitions are used:

Floodwave Arrival Time: Time in hours from when the breach discharge exceeds 0.01 cfs or cms to when the floodplain grid element flow depth exceeds 0.1 ft or 0.3 m. If the grid element has a channel assignment, the time when the channel flow depth becomes one foot higher than the base flow (when breach discharge > 0.01 cfs or cms) is reported.

Flood Time: Time (hours) from when the breach discharge exceeds 0.01 (cfs or cms) to when a given grid element flow depth exceeds 0.5 ft or 0.15 m on the floodplain. If the grid element has a channel assignment, the time to when the flow exceeds the lowest top of bank is reported.

Peak Time: Time (hours) from when the breach discharge exceeds 0.01 (cfs or cms) to when a given grid element flow depth reaches a maximum depth. If the grid element has a channel assignment, the time to when the channel flow reaches a maximum depth is reported.

Deflood Time: Time (hours) from when the flood arrives at a grid element (flow depth exceeds 0.5 ft or 0.15m) to the time when the flood recedes to a depth lower than 0.5 ft or 0.15 m. If the grid element has a channel assignment, the Deflood Time is the time recorded from when the channel flow goes overbank for the lowest top of bank to when water surface recedes to below that lowest top of bank elevation.

Max WS: The maximum water surface elevation for a given floodplain grid element is reported. If a channel is assigned to the grid element, the

maximum water surface elevation for either the channel or the floodplain is reported.

FLOODWAY.OUT

FLOODWAY.OUT is written when IFLOODWAY = 0. This file lists the grid element and the maximum floodplain water surface elevation. Following the base flood simulation in which FLOODWAY.OUT is written, the then user sets IFLOODWAY = 1 and assigns a value for ENCROACH in CONT.DAT. For a floodway simulation, the model reads FLOODWAY.OUT and does not share discharge between floodplain elements until the computed water surface in FLOODWAY.OUT plus the ENCROACH value is exceeded for a given grid element.

FPINFILTRATION.OUT

The total infiltration (ft or m) at the end of the simulation for each floodplain element is written to this file with grid element x- and y-coordinates.

FPLAIN.RGH

This file contains the final Manning's n-value changes for the floodplain grid elements. The maximum and final Manning's n-values are reported in the ROUGH.OUT. If the changes are acceptable, FPLAIN.RGH can be renamed to FPLAIN.DAT for the next FLO-2D flood simulation. This automates the spatial adjustment of n-values for floodplain elements that exceed the limiting Froude number.

FPSEDSIZE.OUT

The initial and final sediment size distribution for the floodplain grid element is written to this file.

HYCHAN.OUT

This channel hydraulics output file contains a hydrograph for each channel element and includes the time, elevation, depth, velocity, discharge and sediment concentration. The maximum discharge and stage are also listed with their times of occurrence. Additional channel flow hydraulic and geometry parameters listed in the output file include: flow area, hydraulic radius, wetted perimeter, top width, width-to-depth ratio, energy slope, and bed shear stress.

HYCROSS.OUT

The output interval time, top width, depth, velocity and discharge are listed for each cross section. The discharge passing the cross section of grid elements is compiled as a hydrograph. The cross section maximum discharge and the individual grid elements are written to the CROSSMAX.OUT file.

HYDROSTRUCT.OUT

The discharge hydrograph from any hydraulic structure is presented in this output file.

HYSTREET.OUT

If a floodplain cross section coincides with a street, the street flow hydrograph for the grid element that is coincidental to the street and the cross section will be recorded in this file.

IMPACT.OUT

This file displays the impact force per linear foot of structure on the floodplain. The data is presented by grid element number, coordinates and force per linear foot or meter.

INFILHY.OUT

The hydraulic conductivities are listed in this file to view the spatial variation. This file contains grid element number, x- and y-coordinates and floodplain hydraulic conductivity.

INTERGWS.OUT

Maximum floodplain water surface elevations based on the grid element elevation in the TOPO.ORI. Values less than TOL are set to zero. Only grid elements and maximum water surface elevations are listed; no coordinates are included.

LEVEE.OUT

The LEVEE.OUT file contains a list of the grid elements with a levee that failed. Failure width, failure elevation, discharge from the levee breach and the time of failure occurrence are listed.

LEVEEDEFIC.OUT

The levee freeboard deficit is listed in this file. Five levels of freeboard deficit are reported:

0 = freeboard > 3 ft (0.9 m)
1 = 2 ft (0.6 m) < freeboard < 3 ft (0.9 m)
2 = 1 ft (0.3 m) < freeboard < 2 ft (0.6 m)
3 = freeboard < 1 ft (0.3 m)
4 = levee is overtopped by flow.

LEVOVERTOP.OUT

The discharge hydrograph overtopping the levee within the grid element is reported in this file. All the levee grid elements being overtopped are reported. The discharge is combined for all the potential levee overtopping directions for the grid element.

MAXQBYDIR.OUT

This output file lists the maximum floodplain grid element discharge by the eight flow directions and the time of occurrence.

MAXQHYD.OUT

This output file lists the maximum floodplain grid element discharge and the associated hydraulics including the maximum discharge flow direction, water surface elevation, flow depth, velocity, and the maximum resolved discharge and the corresponding flow direction.

MAXQRESOLVED.OUT

The maximum discharge resolved by flow direction listed for all eight flow directions regardless of the time of occurrence are reported to this file. The resolved flow direction maximum discharge includes the sum of the primary flow direction and the two diagonal flow directions.

MAXWSELEV.OUT

Similar to DEPTH.OUT, this file contains grid element number, x-coordinate, y-coordinate, and the maximum water surface elevation of either the floodplain or channel.

MULTCHN.OUT

The multiple channel routine routes the overland flow between grid elements as concentrated channel flow (i.e. rill and gully flow). For grid elements specified for multiple channel flow, overland flow only occurs within the grid element and the flow between the elements is conveyed as gully flow. Once the flow enters the multiple channels, the channel will enlarge to contain the flow. This occurs when the flow depth exceeds the specified

channel depth. The channel increases by a specified incremental width. After the peak discharge has passed and the flow depth is less than one foot, the channel width will decrease until it reaches the original width. MULTCHN.OUT identifies multiple channel revisions including the maximum width, final width and the original width for each grid element. The file has the following format:

MULTCHN.OUT File Example

MULTIPLE CHANNEL WIDTH REVISIONS

Node	Max Width	Final Width	Orig. Width
2	425.	384.	50.
28	114.	20.	20.
29	169.	89.	20.
31	15.	5.	5.
55	218.	13.	10.
56	181.	10.	10.
57	92.	10.	10.
58	41.	10.	10.
59	50.	5.	5.
85	437.	117.	5.

OUTNQ.OUT

The OUTNQ.OUT file contains a summary of the maximum discharge, time of peak and the discharge hydrograph for each floodplain outflow element.

OVERBANK.OUT

When the flow exceeds bankfull discharge and begins to inundate the floodplain, the channel grid element and time of overbank flood occurrence are written to this file.

REVISED_RATING_TABLE.OUT

This file reports suggested revisions to hydraulic structure rating tables based on the inflow discharge to the hydraulic structure inlet floodplain or channel element. These revisions are usually the result of the rating table being created with low n-values in a HEC-RAS model or because the rating table has insufficient low depth stage-discharge pairs.

ROUGH.OUT

The ROUGH.OUT file tracks the changes in the Manning's n-values. The user specifies a maximum Froude number for overland, channel and street flow. When the computed Froude number exceeds the specified maximum value for a given grid element, the n-value for that grid element is increased by 0.001. During the falling limb of the hydrograph when the Froude number drops below 0.5, the n-value is decreased by 0.0005 until the original n-value is reached. The maximum, final and original n-values for floodplain, channel and street are listed in ROUGH.OUT by grid element. A portion of a ROUGH.OUT file follows:

SURFAREA.OUT File Example

TABLE OF GRID ELEMENT SURFACE AREAS

UNMODIFIED GRID ELEMENT SURFACE AREA (SQ. FT) = 250000.

NODE	ARF-REDUCED AREA	CHANNEL AREA	STREET AREA	MULT CHANNEL AREA	OVERLAND AREA	MULT CHANNELS (0-NO, 1-YES)
1	175000.	0.	0.	0.	175000.	0
2	175000.	0.	0.	0.	175000.	0
3	175000.	109546.	0.	0.	98318.	0
4	175000.	0.	0.	0.	175000.	0
5	175000.	0.	0.	0.	175000.	0
6	175000.	125790.	0.	0.	86947.	0
7	175000.	0.	0.	0.	175000.	0

ROUGH.OUT File Example

FLOODPLAIN MANNINGS N CHANGES
FP = FLOODPLAIN, MC = MULTIPLE CHANNELS, ST = STREETS

	NODE	MAX n	FINAL n	ORIG. n
FP	2454	0.069	0.065	0.065
FP	2667	0.080	0.065	0.065
FP	2711	0.092	0.065	0.065

SORTED BY MAXIMUM n-VALUE:

NODE	MAX n	TIME (HRS)
2711	0.092	3.074
2667	0.080	3.134
2454	0.069	3.166

CHANNEL MANNINGS N CHANGES

NODE	MAX n	FINAL n	ORIG. n
2985	0.089	0.080	0.080
2968	0.095	0.091	0.071
2946	0.091	0.069	0.069
2922	0.107	0.098	0.098
2894	0.116	0.077	0.077
2863	0.082	0.069	0.069

SORTED BY MAXIMUM n-VALUE:

NODE	MAX n	TIME (HRS)
2894	0.116	2.859
2666	0.109	3.079
2922	0.107	2.479
2830	0.107	2. 700
2968	0.095	2.305
2946	0.091	3.276

SEDCHAN.OUT

The sediment transport routine will compute scour and deposition in the channel. In this output file, the x- and y-coordinates, maximum deposition, maximum scour and final bed elevation change (ft or m) are reported by channel element.

SEDCONSERV.OUT

The sediment transport conservation summary is listed by output interval.

SEDFP.OUT

Similar to the SEDCHAN.OUT file, the floodplain scour and deposition are reported in the SEDFP.OUT file. The x- and y-coordinates, maximum deposition, maximum scour, bed elevation change (ft or m) and max water surface elevation for the floodplain surface are reported by grid element.

SEDTRAN.OUT

The sediment transport capacity (cfs or cms) computations for each of the eleven sediment transport equations are listed by output interval in this file.

SPECENERGY.OUT

The specific energy is the sum of the depth plus the velocity head. This file lists the maximum specific energy (ft or m) for a floodplain grid element and includes grid element number, grid element x- and y-coordinates and maximum specific energy.

STATICPRESS.OUT

The spatially variable static force per linear foot for each floodplain element is presented is this file by grid element number, x- and y-coordinates and force per linear foot or meter.

STREET.RGH

This file lists the final changes to Manning's n-values for the street grid elements. The maximum and final Manning's n-values are reported in the ROUGH.OUT file. If the n-value changes are acceptable, STREET.RGH can be renamed to STREET.DAT for the next FLO-2D flood simulation. This automates the spatial adjustment of n-values for floodplain elements that exceed the limiting Froude number.

STREET.OUT

Similar to DEPTH.OUT, this file contains the street element x- and y-coordinates and the maximum street flow depth.

STRELEV.OUT

Final street elevations used in the model simulation are listed in this file.

SUMMARY.OUT

This file lists the volume conservation summary table including the simulation output time interval, the minimum timestep and flood volume conservation. It also reports the inflow hydrograph, rainfall, infiltration loss, and outflow and storage volumes. Review the volume conservation accuracy and the final distribution of volume in this file.

SUPER.OUT

When the floodplain or channel flow is supercritical, the grid elements with supercritical flow, the Froude number, and time of occurrence are written to this file. This file requires that the variable SUPER = 1 in the CONT. DAT file.

SURFAREA.OUT

The SURFAREA.OUT lists the available flow surface area in each grid element. The area reduction factors (ARF) remove a portion of the surface area of a grid element to account for buildings or other features that occupy the flow surface area. Channels, streets and multiple channels also require a portion of the floodplain surface. The remaining floodplain surface area is reported. At the end of the file, the maximum area of floodplain inundation (including the channel surface area) for the entire grid system is listed by output time interval. This can be an informative data file for the user. If the available flow surface area is less than 5% of the original grid element surface, the FLO-2D simulation will be terminated with error messages. The SURFAREA.OUT file enables a review of the surface area distribution between the various components. This file has the following format:

SWMM.RPT

The results of the SWMM modeling of the storm drain flow routing is written to this file. The EPA SWMM model generates this file. It is ex¬tensive and contains discharge hydrographs for every drain inlet, outlet and conduit.

SWMMQIN.OUT

The discharge hydrograph (time and discharge pairs) into each storm drain inlet as computed by the FLO-2D model for exchange to the SWMM pipe routing model is reported in this file. Each inlet has a discharge hydrograph with a output interval TOUT timestep.

TIMDEP.OUT

This file contains grid element, flow depth, velocity and velocity direction x and y and water surface elevation for each floodplain grid element at the user specified time intervals (TIMTEP in CONT.DAT). This file is also

required for a time-lapse simulation in the MAXPLOT and MAPPER++ post-processor programs.

TIME.OUT

The timestep is controlled by the numerical stability criteria. When the stability criteria are exceeded for a particular grid element, the timestep is decreased. The grid elements with the highest number of timestep decreases are written to the TIME.OUT file. This file can be reviewed to determine if a specific floodplain, channel or street node is consistently causing the timestep decrease and what stability criteria is frequently being exceeded. If one grid element has caused significantly more timestep decreases than the other grid elements, then its attributes and the attributes of the contiguous grid elements should be carefully reviewed. An example of the TIME.OUT file is shown below.

TIME.OUT File Example

TIMESTEPS WERE DECREASED FOR THE FOLLOWING NODES:

GRID ELEMENT	NUMBER OF TIMESTEP DECREMENTS	PERCENT CHANGE IN DEPTH	CFL STABILITY CRITERIA	DYNAMIC WAVE STABILITY CRITERIA
FLOODPLAIN NODES				
4116	40552	0	40552	0
4476	23725	0	23725	0
...				
CHANNEL NODES				
197	1892	0	1	891
229	1467	0	4	1463
...				

TIMEONEFT.OUT

For dam or levee breach this file reports the grid element number, the x- and y-coordinates and the initial time to one foot of depth. The time to one foot of depth can be plotted in MAPPER++.

TIMETOPEAK.OUT

For dam or levee breach this file reports the grid element number, the x- and y-coordinates and the time of occurrence of the maximum depth. This time to maximum depth can be plotted in MAPPER++.

TIMETWOFT.OUT

For dam or levee breach this file reports the grid element number, the x- and y-coordinates and the initial time to two feet of depth. The time to two feet of depth can be plotted in MAPPER++.

VELOC.OUT and Related Velocity Output Files

These files are similar to the DEPTH.OUT file. These files contain the x- and y-coordinates and maximum velocities and can be viewed with the MAXPLOT or MAPPER++ program.

The velocity output files include:

STVEL.OUT - Maximum street flow velocity;
STVELDIR.OUT - Flow direction of the maximum street flow velocity;
VELFP.OUT - Maximum floodplain flow velocity;
VELOC.OUT - Maximum channel flow velocity;
VELCHFINAL.OUT - Final channel flow velocities.;
VELDIREC.OUT - Flow direction of the maximum floodplain flow velocity.
FINALVEL.OUT -Flow velocity at the end of the simulation.
FINALVELDIR.OUT - Flow maximum velocity direction at the end of the simulation.

VELTIMEC.OUT

This file lists the channel element number, maximum channel velocity and the time of occurrence. It is sorted from highest to lowest velocity so that an examination of the first several lines of output data will determine if there are any unreasonably high maximum channel velocities.

VELTIMEFP.OUT

Lists the floodplain element number, maximum channel velocity and the time of occurrence. This file is sorted from highest to lowest velocity so that an examination of the first several lines of output data will determine if there are any unreasonably high maximum floodplain velocities.

VELTIMEST.OUT

Lists the street element number, maximum channel velocity and the time of occurrence. This file is sorted from highest to lowest velocity so that an examination of the first several lines of output data will determine if there are any unreasonably high maximum street velocities.

WSTIME.OUT

If WSTIME.DAT file is created, the WSTIME.OUT file will be generated listing the channel element number, time of the measured water surface elevation, measured water surface elevation at stated time, predicted water surface elevation at stated time, difference between the water surface elevations and the cumulative difference between the measured and predicted water surfaces.

XSECAREA.OUT

When the channel cross section option is invoked for channel routing, the channel geometry data is written to this file. It includes: grid element, flow area, top width and wetted perimeter for the lowest top of bank (bankfull flow).

XSEC.OUT

This file is created by the sediment transport option (ISED = 1 in CONT. DAT and ISEDN = 1 for a channel segment in CHAN.DAT) for channel cross section data. It contains the final cross section bed elevations after scour and deposition have been computed.

CHAPTER 6

POST-PROCESSOR PROGRAMS

There are four post-processor programs: MAPPER++, MAXPLOT, HYDROG and PROFILES that display the output data graphically. MAPPER++ and MAXPLOT display the flood inundation depth and velocity plots. Channel flood simulations or floodplain cross sections are required to view hydrographs in HYDROG. A channel model is also required to view the channel water surface profiles in PROFILES. These programs can be initiated by clicking on their names in the File pull down menu in the GDS or by copying the executable (*.EXE) file into the project subdirectory and double clicking on the program name in a bowser. You can also put the icons on the desktop.

6.1 HYDROG

Channel output hydrographs and floodplain cross section hydrographs can viewed with the HYDROG program. It displays the hydrograph for every channel element in the system. HYDROG will interactively display the channel hydrograph. It will also list the average channel hydraulic data for various reaches of river. Gaging station hydrograph data can be plotted along with the FLO-2D predicted hydrograph by creating the optional HYDRO.DAT file in the following format:

Instructional commands for HYDROG follow:

HYDRO.DAT File Descriptors

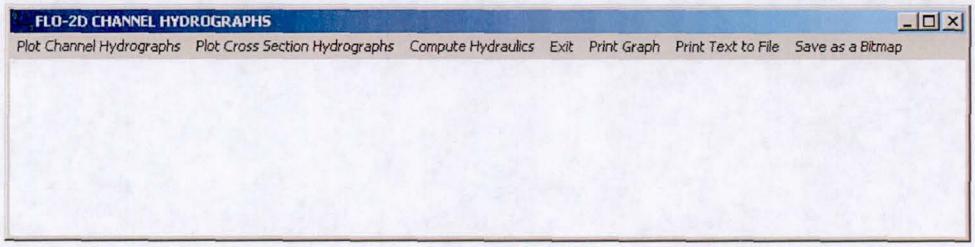
3	Line 1:	Number of gaging station with hydrograph data
Haynor	Line 2:	Name of gaging station (10 letters)
13160 251	Line 3:	Channel grid element and # of hydrograph pairs
0.00 1287.85	Line 4:	Hydrograph pairs time(hours) discharge (cfs)
1.00 1285.47	Line 4:	Hydrograph pairs time(hours) discharge (cfs)

Notes:
Line 2 - 4: These lines are repeated for each gaging station.

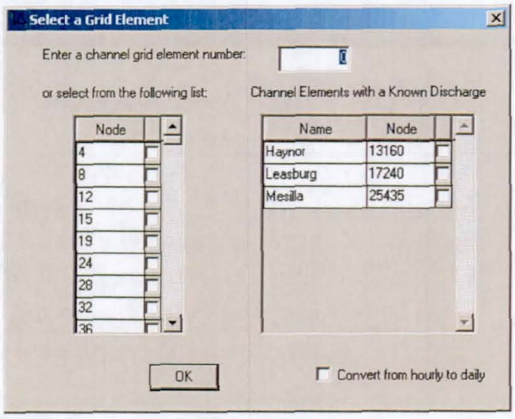
HYDRO.DAT File Example

```
Haynor
13160    251
0.00     1287.85
1.00     1285.47
2.00     1295.01
3.05     1302.20
...
```

After opening HYDROG, click on either 'Plot Channel Hydrographs' or 'Compute Hydraulics' in the Main Menu shown below:

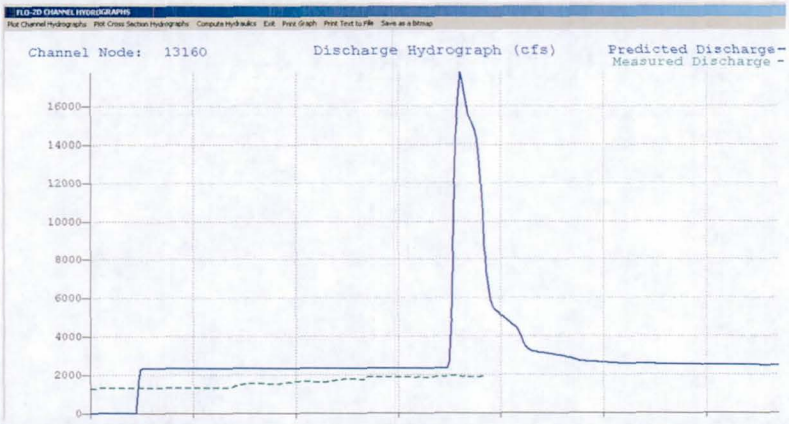


If you click on 'Plot Channel Hydrographs' a dialog box appears to select either a channel segment or element:

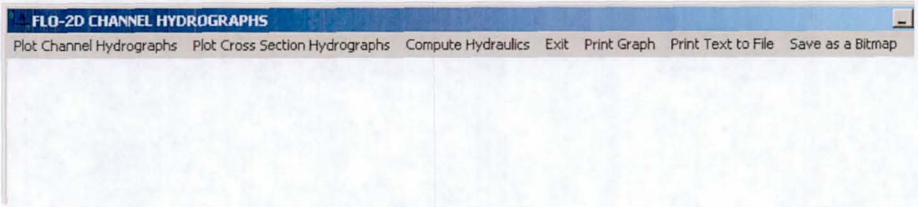


> **Hint:**
> From the Main Menu, other options are to save the plotted hydrograph as a bitmap image, send the hydrograph to a printer, or write the hydrograph to an ASCII file.

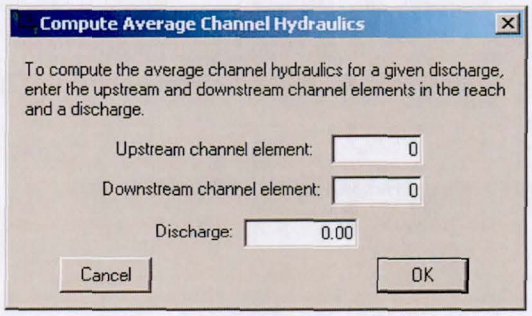
After selecting the channel element and clicking 'OK', the hydrograph is plotted as shown in the following figure. Use the dialog box in the upper right portion of the screen to select another channel element or to return to the channel element list or main menu.



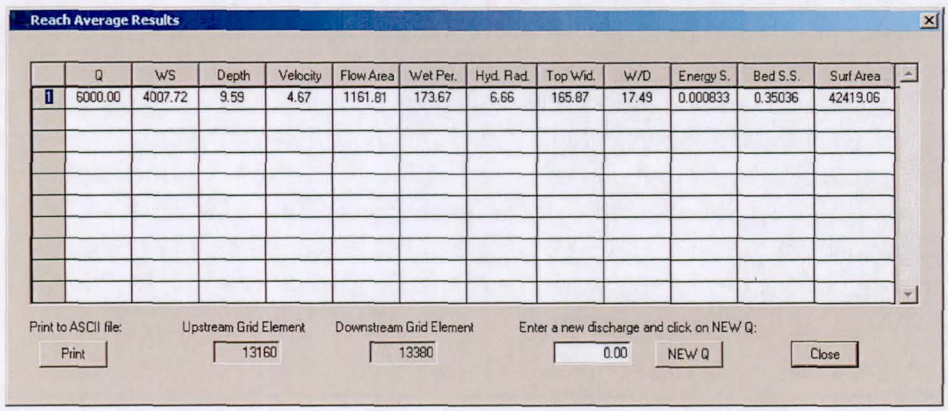
Similar hydrographs can plotted for floodplain cross section selected in the FPX-SEC.DAT. If this file exists, then the crosss section hydrograph for the selected cross section elements and flow direction will plotted using the second command on the Main Menu bar.



If the 'Compute Hydraulics' is selected from the Main Menu, the following dialog box is displayed:



After filling the three data fields in the dialog box (including the desired discharge for computing the average channel hydraulics, mouse click 'OK' to display the following table:



	Q	WS	Depth	Velocity	Flow Area	Wet Per.	Hyd. Rad.	Top Wid.	W/D	Energy S.	Bed S.S.	Surf Area
1	6000.00	4007.72	9.59	4.67	1161.81	173.67	6.66	165.87	17.49	0.000833	0.35036	42419.06

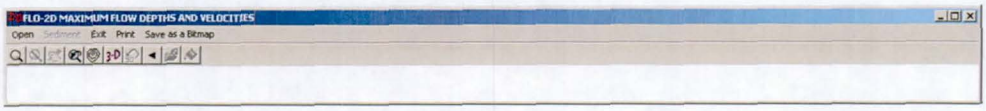
This table displays the average discharge weighted hydraulic conditions for the given discharge between the two channel elements (inclusive). From this dialog box, you can select a new discharge and add to the table or print this table to an ASCII file (HYDR.OUT).

6.2 MAPPER ++

MAPPER++ is a post-processor program that creates high resolution maps and plots of the FLO-2D model results including area of inundation, time variation of hydraulic variables, maximum water surface elevations, duration of inundation, impact force, static pressure, specific energy, sediment scour or deposition and oth-

ers. MAPPER++ is the primary post-processing program for viewing the FLO-2D simulation results. Three types of plots can be generated:

- Grid element plots where each element is assigned a color depending on the value of the selected plot variable.
- Line and shaded contour maps based on the grid element values.
- DTM point depth plots to generate detailed flow depth contour maps based on grid element water surface elevations and DTM point ground elevations.



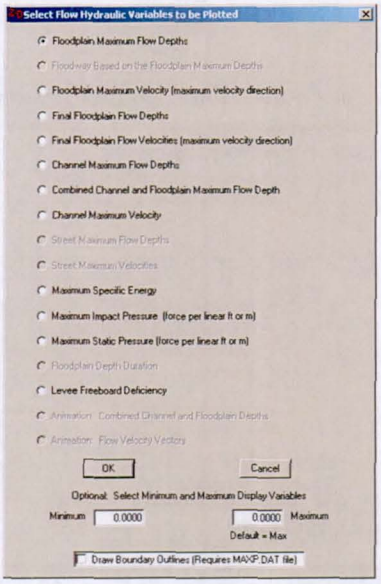
The MAPPER++ manual describes the commands and tools and provides instruction.

6.3 MAXPLOT

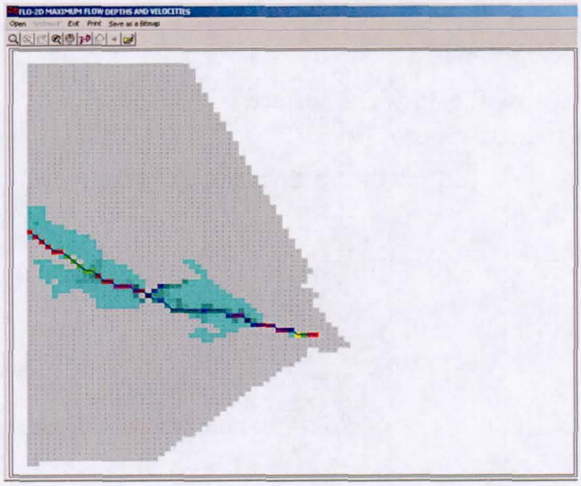
The MAXPLOT program is a simple graphical tool to display the grid element maximum depths and velocities. MAXPLOT is a simple alternative to MAPPER++ that quickly displays plots of the maximum floodplain and channel depths, maximum street velocity, final floodplain depths and others. It is faster than MAPPER++ but has less graphical resolution and fewer display options. Use MAXPLOT when you just want to get a quick overview of predicted flow depths and velocities. By zooming in on a given plot, the grid element number, maximum flow depth or velocity and the maximum water surface elevation can be viewed. The tool bar has options for view extents, previous view, pan, a coarse flood contour and 3-D plot and an option to save the view as a bitmap.

MAXPLOT can be initiated by copying the program to the project folder and double clidking it. After opening MAXPLOT a blank screen appears with a Main Menu:

Click on 'Open' to display the following the dialog box:



Activate one of the plots listed in the dialog box above by clicking on the radio button in front of the plot option and clicking the 'OK' button. You can also set limits on the minimum and maximum depths or velocities to display. The following plot displays the combined channel and floodplain maximum flow depth for the Monroe project example.



MAXPLOT also has an option to plot a floodplain boundary such as the area of inundation, level location, or other linear boundary (see the bottom of the above dialog box). The MAXP.DAT file lists the boundary line locus of points must be prepared to display the boundary. A series of line boundaries can be prepared.

Each set of boundary points will constitute a separate line. The optional MAXP. DAT file format is:

Line 1. Number of boundary units. Each boundary set constitutes a line.

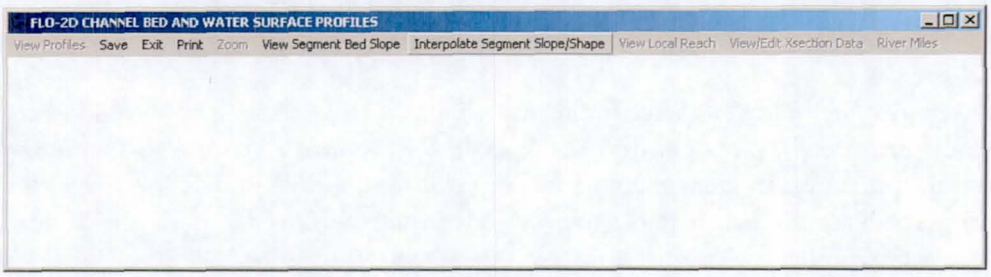
Line 2. Number of points in each boundary set.

Line 3 to end. x- and y-coordinates of each point listed in columns.

Repeat Lines 2 and 3 for each boundary line set of points.

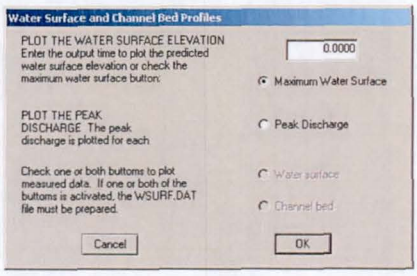
6.4 PROFILES

The PROFILES program serves the dual purpose of being a pre- and post-processor program. As a post-processor program, it will display a channel water surface and bed elevation for any FLO-2D simulation output interval. In order to view the predicted water surface elevation in PROFILES, it is necessary to run a FLO-2D

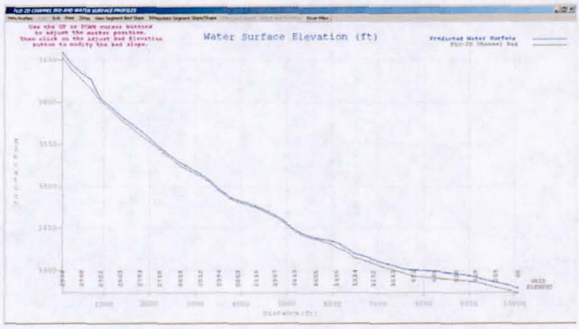


channel simulation first. The PROFILES program has zoom and print options to assist in reviewing the results.

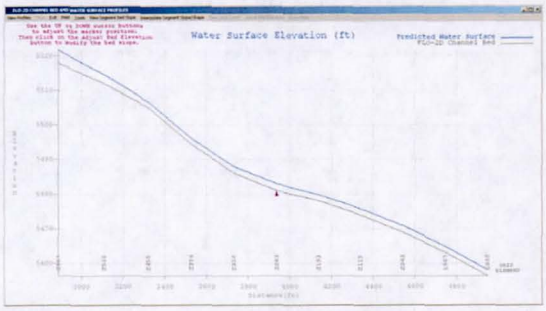
To view the predicted water surface profiles, click on 'View Profiles' in the Main Menu and a dialog box appears:



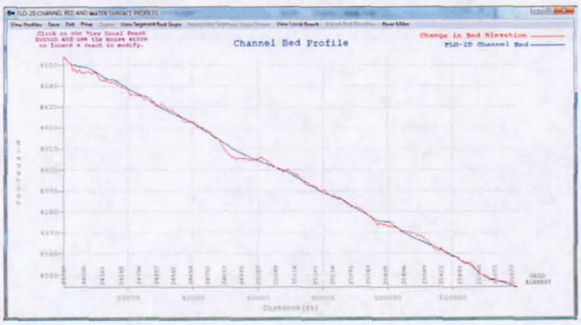
To view the predicted maximum water surface elevation profile, click on the radio button labeled 'Maximum Water Surface' and click 'OK'. You can plot the water surface at any output interval by entering the time in the text box in the upper right corner. The peak discharge can also be plotted as a function of the channel distance. To plot the surveyed or measured, the WSURF.DAT must be prepared. The file format is presented at the end of this section of the manual.



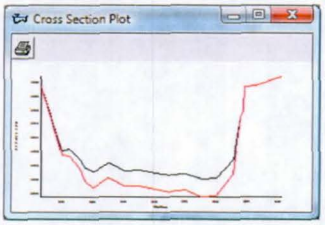
There are several options on the Main Menu. You can zoom in on given river reach, print the image or label the distance along the channel in river miles. The zoom view is shown in the following figure:



If sediment transport has been simulated, PROFILES will plot the final bed elevation and the cross section geometry changes associated with either scour or deposition. The image below displays sediment deposition of approximately 1 ft upstream of a bridge in the Rio Grande.



Non-uniform sediment distribution on the channel bed can be viewed when the channel flow is simulated. The cross section plot below displays the final cross section elevations in red compared to the cross section elevations at the start of the flood simulation shown in black. This image can be expanded to full view.



The user has an option in the water surface dialog box of plotting surveyed water surface and bed elevations along with the predicted values. To plot the surveyed water surface or channel bed elevation, the WSURF.DAT file must be created in the following format:

WSURF.DAT File Descriptors

2045	Line 1: # of channel elements with a surveyed ws elev.
4 4152.22	Line 2: Grid Element WS elevation
8 4151.84	Line 2: Grid Element WS elevation
...	

Notes:
Line 2: This line is repeated for each channel element with a surveyed ws elevation.

WSURF.DAT File Example

```
2045
   4   4152.22
   8   4151.84
  12   4151.69
  15   4151.55
  19   4151.41
  ....
```

Optional WSURF.DAT file format:

Please note that PROFILES also has options for editing the channel bed slope and thalweg flow depth and for interpolating the slope and cross section geometry for the cross section option. Refer the section on Pre-Processor Programs for a discussion on these features.

The surveyed water surface can also be compared directly with the FLO-2D computed water surface in the WSTIME.OUT (see file description in the output file section) by creating a WSTIME.DAT file. The WSTIME.DAT file format is as follows:

WSTIME.DAT File Descriptors

49			Line 1:	# of channel elements with a surveyed ws elev.
117632	4658.95	240	Line 2:	Grid Element WS elevation Time
117928	4655.80	240	Line 2:	Grid Element WS elevation
...				

Notes:
Line 2: This line is repeated for each data set.

WSTIME.DAT File Example

```
49
117632   4658.95   240
117928   4655.80   240
119882   4652.28   240
120580   4650.36   240
120915   4648.52   240
....
```

The WSTIME.OUT file will contain:

Channel element number., time of survey (hrs), surveyed water surface elevation, computed water surface elevation, difference between the surveyed and computed water surface and cumulative difference between the surveyed and computed water surface elevations.

CHAPTER 7

DEBUGGING AND TROUBLE SHOOTING THE DATA FILES

7.1 BASIC TROUBLE SHOOTING GUIDELINES

Start simple and build in the flood model detail. After the required data files have been prepared, run a basic overland flood simulation. Review the results and if you have problems consult the troubleshooting tips found in this chapter.

The first step in debugging the data files after attempting a FLO-2D simulation is to review the ERROR.CHK file. All the data errors recognized by the model are reported in this file. One of the most common errors is missing data that will invoke an end-of-file error statement to the screen. End-of-file messages are reported to the ERROR.CHK file. This error occurs when the model is searching for more data than is in the data file. Another common error is to activate a component or process switch without preparing the required data file. For example, an error will occur if MUD = 1 in the CONT.DAT file, but the data file SED.DAT is not prepared or if the sediment concentration in INFLOW.DAT is left out.

One data error that is difficult to locate is the array allocation violation where the array index number may be zero or larger than the assigned value. When this type of error is encountered, the FLO-2D model is terminated with an error message without indicating the file location or line entry of the error. To locate the data error, simplify the simulation and turn off all of the components and add them back to the model one at time until the error occurs again. You can reset SIMUL in CONT.DAT to the model time just after the error occurred to reduce time to debug the model. If your attempts to debug an error are ineffective, send a zipped copy of the data files to FLO-2D (support@flo-2d.com) along with brief description of the problem.

You can create a set of backup data files to debug the model. Set IBACKUP = 1 in the CONT.DAT file. These backup files replicate your data files and will indicate if the computer is reading the data files correctly. The backup file should be identical to the original data file except for spacing. If the program terminates before reaching the first output interval timestep, there is probably an error in the data files. Start by checking the *.BAC files one by one. If one of the files is not complete, this may be the location of the data error.

Review the following files to analyze volume conservation problems: SUMMARY.OUT, CHVOLUME.OUT, CHANGRID.OUT, CHANMAX.OUT, TIME.OUT, BASE.OUT, ROUGH.OUT, CHANNEL.CHK, and SURFAREA.OUT. See the 'Pocket Guide' for further troubleshooting tips involving volume conservation, sticky grid elements listed in the TIME.OUT file, and numerical surging. The instructional comments at the end of each data file description in this manual contains a number of guidelines to assist the user in creating or checking the data files.

7.2 LIST OF COMMON DATA ERRORS

A list of the most common errors associated with running FLO-2D is presented below and a table for troubleshooting runtime errors follows the list. Whenever an error is encountered, refer to the ERROR.CHK file first.

TABLE 7.1. LIST OF MOST COMMON DATA ERRORS

1.	Missing data entries. Insufficient data was provided to the model.
2.	Switches were activated without the corresponding data or files (for example, see MUD, ISED, etc., in the CONT.DAT file).
3.	There was missing or additional lines in a data file when switch is activated. Observe the *** Notes: *** in the file descriptions.
4.	Percentages were expressed as a number instead of a decimal. See the description of XCONC in CONT.DAT or the HP(I,J,3) variable in INFLOW.DAT.
5.	The IDEPLT grid element was improperly assigned in INFLOW.DAT for the graphics mode.
6.	Channel infiltration switch INFCHAN was not 'turned on' in the INFIL.DAT file.
7.	Either one or both of channel and floodplain outflow elements were not assigned for a given grid element.
8.	The street width exceeded the grid element width.
9.	The array size limitation for a variable was exceeded.

TABLE 7.1. LIST OF MOST COMMON DATA ERRORS
10. The available floodplain surface area was exceeded by assigning channels, streets, ARF's and/or multiple channels with too much surface area. Review the SURFAREA.OUT.
11. The rainfall variable R_DISTRIB data was entered as total cumulative rainfall instead of the percentage of the total rainfall (range 0.0 to 1.0).
12. The ISEDN switch for channel sediment transport was not 'turned on' in the CHAN.DAT file for the channel segment.

7.3 RUNTIME ERRORS

The following guidelines are provided to assist the user to resolve runtime errors. If the program stops executing after several output time intervals but has not reached the prescribed simulation time, the problem grid element(s) may be listed at the end of the BASE.OUT file. It may also be listed in the TIME.OUT file and it may appear in a dialog box that appears on the screen when the simulation terminates. The user should review the following data before revising the stability criteria:

- Review the problematic grid element elevation and Manning's n-value for the listed grid element as well as the contiguous 8 grid elements neighbors. This can be done in the GDS program.
- Check the channel data to determine if the grid element contains a channel and then check the relationship between the channel bed and bank elevations and the floodplain elevation within the grid element.
- Review the floodplain area available within the grid element including loss of floodplain area due to channels, streets, ARF's or multiple channels in the SURFAREA.OUT file.

Most volume conservation and numerical stability problems are associated with channel flow. When constructing a channel system, it is often necessary to fabricate cross section geometry, estimate roughness or adjust channel bed slopes. Mismatched channel morphology parameters with an appropriate roughness are the primary source of numerical stability problems. To compute smoother hydraulics between two channel grid elements, adjust the bed slope, cross section flow area or roughness values. Try to avoid abrupt changes in cross sections geometry from one channel element to another. The channel flow area should make a gradual transition from a wide, shallow cross section to a narrow deep cross section. An actual cross section transition may occur over several channel grid elements. Adjust the channel geometry so that the maximum change in flow area between channel elements is less than 25%. To address channel problems, consider the following measures:

- Increase the roughness in wide, shallow cross sections and decrease the roughness in narrow deep channel grid elements.
- Reduce the difference between the cross section areas. Avoid abrupt cross section transitions between channel elements. Adjust the channel cross section geometry in the PROFILES. Using PROFILES, you can re-interpolate between surveyed cross sections.
- Review and adjust the bed slope with the PROFILES program.
- Select a longer channel length within the channel grid element.

7.4 TROUBLESHOOTING GUIDE

In addition to the following troubleshooting guide, refer to the 'Getting Started Guidelines' at the begin of this manual and the Pocket Guide to assist you in debugging runtime errors.

Program will not run

- Data errors. Turn off the component switches until the model runs.
- The executable program was damaged. Reload the program or contact technical support.
- The model is not properly licensed. Contact technical support.

Program stops

The model run is terminated before the first timestep or after a few timesteps with data file error indicated on the screen

- Review the ERROR.CHK file or the data file identified by the program error message.
- Review the backup file (*. BAC).
- Review the List of Common Data Errors.

Program stops

The model run is terminated after several timesteps indicating a numerical stability error. The grid element causing the stability error is listed on the screen instability dialog box or at the end of the BASE.OUT file.

Stability criteria were not met.

Review and revise the elevation and roughness data for the indicated grid element. The ROUGH.OUT and TIME.OUT files will help to locate the

problem grid element. Also, check the contiguous grid elements to the problem element in the 8 directions.

Volume conservation

The volume conservation may indicate either a loss or gain of volume. A review of the SUMMARY.OUT and CHVOLUME.OUT will reveal if the volume conservation error is in the channel or on the floodplain. Volume conservation problems are indication of data error.

Discharge surging

Numerical surging (alternating from low to high discharges) is usually associated with channel flow. Floodplain surging can occur but is less common. Review the maximum floodplain velocities in MAXPLOT and in the VELTIMEC.OUT and VELTIMEFP.OUT files. Unreasonable maximum velocities should be addressed. Other files to review for indications numerical surging include CHANMAX.OUT, HYCHAN.OUT, CHANSTABILTY.OUT, TIME.OUT, and ROUGH.OUT files. The hydrograph plots in the HYDROG program may display spikes to indicate surging. It should be noted that surging may occur and and the model could have good volume conservation.

Supercritical flow

Supercritical flow is not necessary a problem, but its occurrence should be limited to conditions where it is expected such as in streets, concrete channels or steep bedrock watersheds. Supercritical flow on alluvial surfaces should be avoided.

Numerical Instability:

The channel surging may be related to numerical instability, abrupt changes in channel geometry, inappropriate slopes, supercritical flow or variable mudflow sediment concentrations. Mismatched slope, flow area and n-values are the most common causes of channel instability. A combination of revisions may improve numerical instability.

- Abrupt changes in slope or severe adverse slope may cause instability. Use the PROFILES program to fix irregular bed slope conditions.
- Review the cross section flow areas over several channel elements in PROFILES. Eliminate any abrupt changes in cross section areas between channel elements. If the surging occurs at low flows, review only the bottom portion of the cross section not the bankfull conditions.

- Decrease the channel Courant number in the TOLER.DAT file. Decrease the Courant number in 0.1 increments until 0.2 is reaches.
- Insufficient floodplain area. Small floodplain surface areas can exacerbate unsteady flow. Review SURFAREA.OUT and increase the available grid element surface area for flood storage.
- Increase the n-values for the grid elements in the vicinity of the surging flow.
- Adjust the floodplain grid element elevations around the problem element.
- Increase the channel length within the grid element.
- The hydraulic structure discharge rating curve or table may be poorly matched with the upstream or downstream channel hydraulics. Review the hydraulic structure rating curve or table and compare the discharge values to those found in the HYCHAN.OUT file for that particular channel element.

Unexpected supercritical flow on alluvial surfaces:

- Adjust the limiting Froude number using the FROUDL variable in the CONT.DAT file or the FROUDC variable in the CHAN.DAT.
- Increase the floodplain or channel roughness values.
- Modify the slope. The grid elevations assigned by the GDS may not be representative of the field condition. Change the grid element elevations to make the channel or floodplain slope more uniform.

Variable mudflow sediment concentration:

- Review the sediment concentration in the inflow hydrographs in the INFLOW.DAT file.
- The relationships for viscosity and yield stress should fall with the research data presented in the reference manual.

FLO-2D simulation runs slow

Review the TIME.OUT file to identify the elements that have caused most of the timestep reductions. Small timesteps are the result of the model continually exceeding the numerical stability criteria for a small group of grid elements. The change in flow depth for a timestep may be too large. One of primary reasons for a slow flood simulation is that the relationship between the discharge flux and grid element surface area is poor. The rate of

change in the discharge may be too high for the selected grid element size. Increasing the grid element size is the best way to fix a very slow model. Other solutions may include:

- Adjust the channel geometry in transition reaches.
- Create a more uniform channel or floodplain slope.
- Revise the roughness values or limit the supercritical flow.
- Reduce the channel width, street width, ARF values or other parameters to increase the floodplain surface area. Review the SURFAREA.OUT file.
- Increase the grid element size (a last resort).

Graphics display is not activated

The graphics display is controlled by three variables: LGPLOT and GRAPTIM found in the CONT.DAT file and IDEPLT in INFLOW.DAT. IDEPLT is the floodplain or channel inflow grid element whose hydrograph will be plotted on the display.

- Set LGPLOT = 2.
- Check the GRAPTIM variable for an acceptable update interval.
- Make sure that the IDEPLT variable matches an inflow grid element has a hydrograph assigned to it in INFLOW.DAT.

The inflow hydrograph does not plot in the graphics display

- No hydrograph is associated with the IDEPLT variable.
- The hydrograph duration is too long. Reduce the hydrograph length.
- The rainfall duration is too long. Reduce the rainfall time.
- Inappropriate peak discharge or total rainfall values distort the scale for hydrograph plot.

Program stops. Excessive flow depths

If flow depths are excessive, then ponding or surging may be occurring.

- Identify the problem element in MAXPLOT or in the end of the BASE.OUT file.
- Check TIME.OUT to determine if the problem element is also causing the model to run slowly.
- Check the elevation of the problem grid element in the FPLAIN.DAT or in the GDS.

- If the depressed element is a gravel pit or some other feature, increase the n-value to decrease the vertical velocity into the pit.

Erratic discharge in the channel elements.

A review of plotted hydrographs in HYDROG or an examination of the CHANMAX.OUT or HYCHAN.OUT files will reveal if the flow discharge between contiguous channel elements is surging with spikes when a consistent rise or fall of the downstream discharge is expected.

Channel surging can be natural phenomena. Rivers can rise and fall over a few tenths of a foot in matter of seconds. During high flow, the variation in discharge associated with stage change ~0.2 ft can be 1,000 cfs or more. Review the numerical surging troubleshooting. If the channel surging is severe, there two conditions to review are:

- Review the channel confluence and make the confluence pairs are properly assigned. See the CONFLUENCE.OUT file.
- The channel grid elements in the CHAN.DAT file may be misidentified.

Erratic flow in the floodplain grid elements.

Erratic flow in the floodplain grid elements is usually the result of errors in the FPLAIN.DAT file. This type of error generally occurs when the user edits the FPLAIN.DAT file manually and adds, subtracts or moves grid elements around. Virtually all erratic flow conditions on the floodplain can be corrected by revisions either to n-values or elevations in the GDS.

Channel extends through another channel element.

The right channel bank assignments are automated in the GDS. The multiple left bank elements can be assigned to the same right bank on a river bend. The line connecting the left and right bank elements cross other bank connecting lines the model will result in an error message reported in ERROR.CHK file.

The channel bank elements can be viewed in the GDS. If there is a problem with the channel outline, simply revised the right bank element. The right bank element can be any grid element if it does not cross another connecting channel bank line. A right bank element can connect to more than one left bank element.

Program stops; identifying one or more grid elements with too little floodplain surface.

The model will report an error in ERROR.CHK if the channel right bank has is too little surface storage area on the floodplain portion of the element. If this problem occurs and the floodplain surface is less than 5%, then there are several solutions:

- Reduce the ARF value, multiple channel area or street area.
- The channel area can be reduced by decreasing the XLEN variable or top width, which is a function of the channel in the natural channels, the side slopes, or the bottom width in the trapezoidal cross section or the width in the rectangular cross section.
- As a last resort the grid element size can be increased, but this requires the re-generation of the grid system.

CADPTS.DAT error

If grid elements are added or deleted to grid system and the FPLAIN.DAT file is revised, the CADPTS.DAT will report errors if it is not also modified. The CADPTS.DAT file must be changed to reflect the revisions in the FPLAIN.DAT file. Use the CHECKER program to verify the accuracy of the revisions in the FPLAIN.DAT file.

HYDRO.DAT File Descriptors

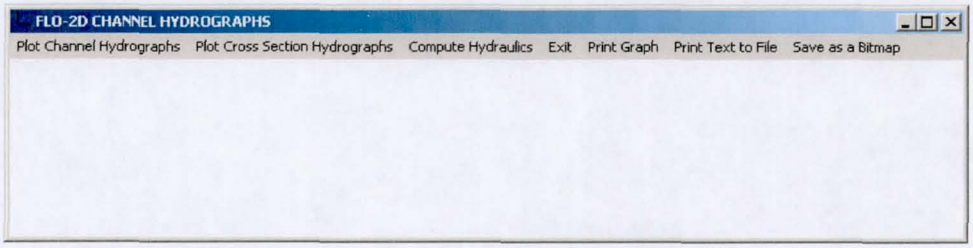
3	Line 1:	Number of gaging station with hydrograph data
Haynor	Line 2:	Name of gaging station (10 letters)
13160 251	Line 3:	Channel grid element and # of hydrograph pairs
0.00 1287.85	Line 4:	Hydrograph pairs time(hours) discharge (cfs)
1.00 1285.47	Line 4:	Hydrograph pairs time(hours) discharge (cfs)

Notes:
Line 2 - 4: These lines are repeated for each gaging station.

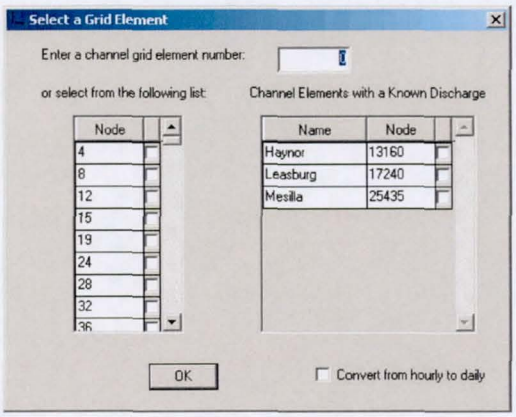
HYDRO.DAT File Example

```
Haynor
13160    251
0.00     1287.85
1.00     1285.47
2.00     1295.01
3.05     1302.20
...
```

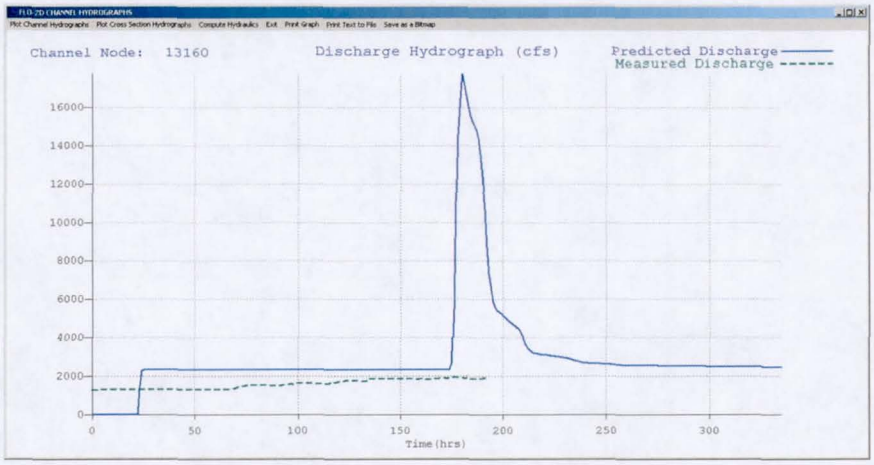
After opening HYDROG, click on either 'Plot Channel Hydrographs' or 'Compute Hydraulics' in the Main Menu shown below:



If you click on 'Plot Channel Hydrographs' a dialog box appears to select either a channel segment or element:

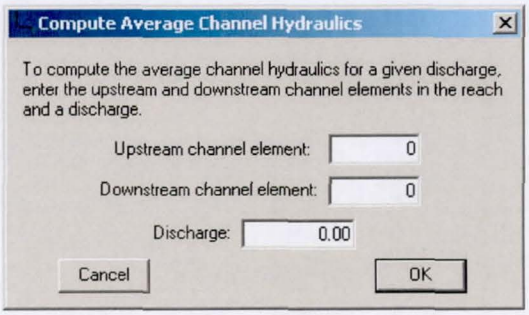


After selecting the channel element and clicking 'OK', the hydrograph is plotted as shown in the following figure. Use the dialog box in the upper right portion of the screen to select another channel element or to return to the channel element list or main menu.



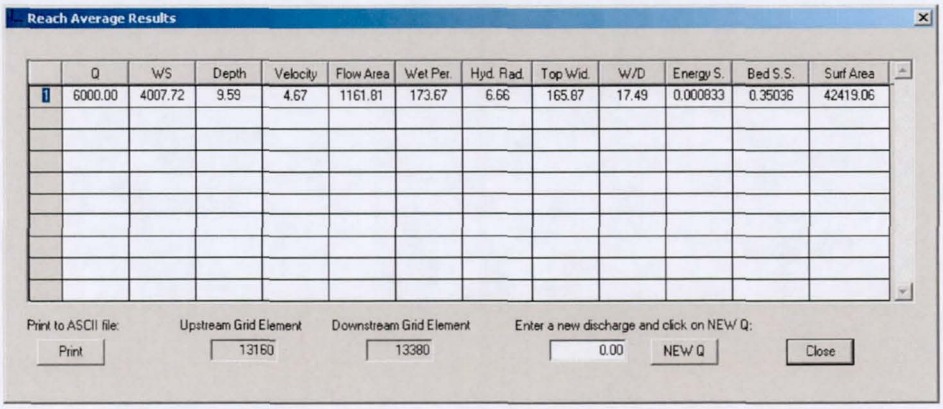
> **Hint:**
>
> From the Main Menu, other options are to save the plotted hydrograph as a bitmap image, send the hydrograph to a printer, or write the hydrograph to an ASCII file.

If the 'Compute Hydraulics' is selected from the Main Menu, the following dialog box is displayed:



Mouse click 'OK' to display the following table:

This table displays the average discharge weighted hydraulic conditions for the given discharge between the two channel elements (inclusive). From this dialog box, you can select a new discharge and add to the table or print this table to an ASCII file (HYDR.OUT).



	Q	WS	Depth	Velocity	Flow Area	Wet Per.	Hyd. Rad.	Top Wid.	W/D	Energy S.	Bed S.S.	Surf Area
1	6000.00	4007.72	9.59	4.67	1161.81	173.67	6.66	165.87	17.49	0.000833	0.35036	42419.06

6.2 MAPPER

MAPPER is a post-processor program that creates maps and plots of the FLO-2D model results including area of inundation, time variation of hydraulic variables, maximum water surface elevations, duration of inundation, impact force, static pressure, specific energy, sediment scour or deposition and others. MAPPER is the primary post-processing program for viewing the FLO-2D simulation results. Three types of plots can be generated:

- Grid element plots where each element is assigned a color depending on the value of the selected plot variable.
- Line and shaded contour maps based on the grid element values.
- DTM point depth plots to generate detailed flow depth contour maps based on grid element water surface elevations and DTM point ground elevations.

The MAPPER manual describes the commands and tools and provides instruction.

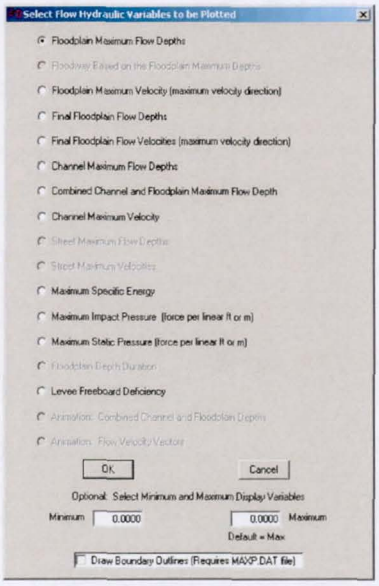
6.3 MAXPLOT

The MAXPLOT program is a simple graphical tool to display the grid element maximum depths and velocities. MAXPLOT is a simple alternative to MAPPER that quickly displays plots of the maximum floodplain and channel depths, maximum street velocity, final floodplain depths and others. It is faster than MAPPER but has less graphical resolution and fewer display options. Use MAXPLOT when you just

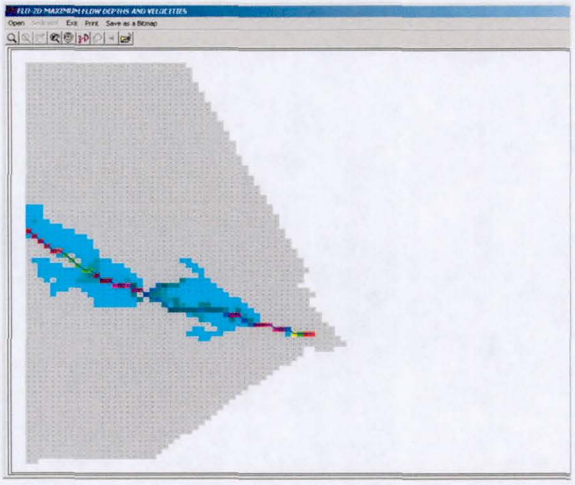
want to get a quick overview of predicted flow depths and velocities. By zooming in on a given plot, the grid element number, maximum flow depth or velocity and the maximum water surface elevation can be viewed. The tool bar has options for view extents, previous view, pan, a coarse flood contour routine, a coarse 3-D view and an option to save the view as a bitmap.

MAXPLOT can be initiated from the GUI Post-processor pull-down menu or it can be started from Explorer by double clicking on the MAXPLOT name. After opening MAXPLOT a blank screen appears with a Main Menu:

Click on 'Open' to display the following the dialog box:



Activate one of the plots listed in the dialog box above by clicking on the radio button in front of the plot option and clicking the 'OK' button. You can also set limits on the minimum and maximum depths or velocities to display. The following plot displays the combined channel and floodplain maximum flow depth for the Monroe project example.



MAXPLOT also has an option to plot a floodplain boundary such as the area of inundation, level location, or other linear boundary (see the bottom of the above dialog box). The MAXP.DAT file lists the boundary line locus of points must be prepared to display the boundary. A series of line boundaries can be prepared. Each set of boundary points will constitute a separate line. The optional MAXP. DAT file format is:

Line 1. Number of boundary units. Each boundary set constitutes a line.

Line 2. Number of points in each boundary set.

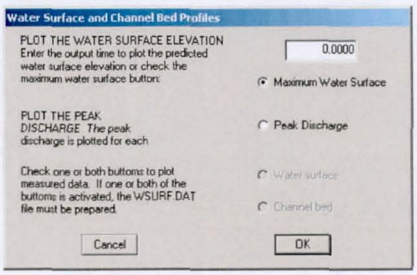
Line 3 to end. x- and y-coordinates of each point listed in columns.

Repeat Lines 2 and 3 for each boundary line set of points.

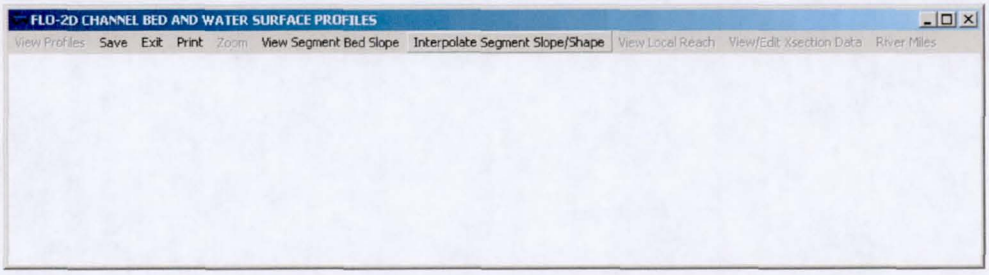
6.4 PROFILES

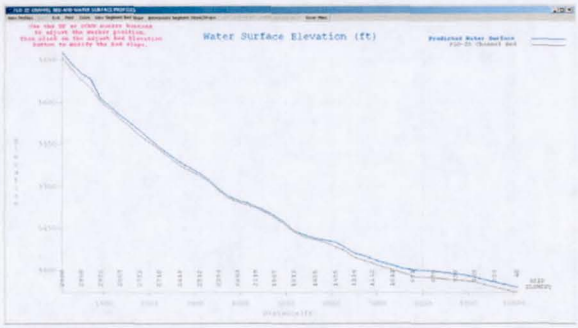
The PROFILES program serves the dual purpose of being a pre- and post-processor program. As a post-processor program, it will display a channel water surface and bed elevation for any FLO-2D simulation output interval. In order to view the predicted water surface elevation in PROFILES, it is necessary to run a FLO-2D channel simulation first. The PROFILES program has zoom and print options to assist in reviewing the results.

To view the predicted water surface profiles, click on 'View Profiles' in the Main Menu and a dialog box appears:

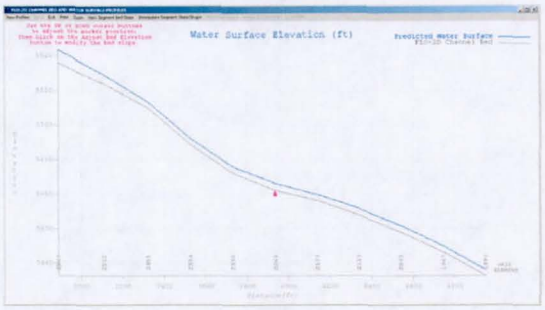


To view the predicted maximum water surface elevation profile, click on the radio button labeled 'Maximum Water Surface' and click 'OK'. You can plot the water surface at any output interval by entering the time in the text box in the upper right corner. The peak discharge can also be plotted as a function of the channel distance. To plot the surveyed or measured, the WSURF.DAT must be prepared. The file format is presented at the end of this section of the manual.

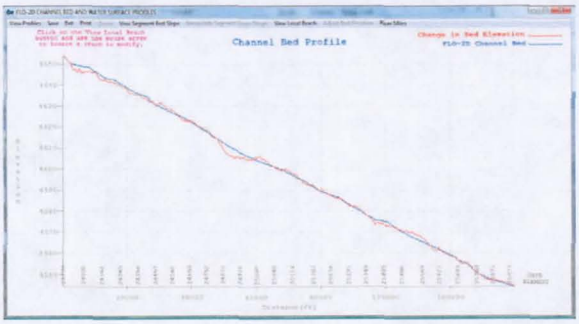




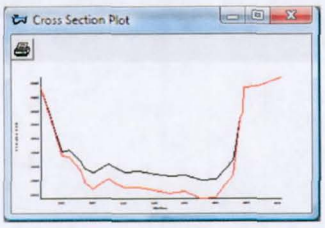
There are several options on the Main Menu. You can zoom in on given river reach, print the image or label the distance along the channel in river miles. The zoom view is shown in the following figure:



If sediment transport has been simulated, PROFILES will plot the final bed elevation and the cross section geometry changes associated with either scour or deposition. The image below displays sediment deposition of approximately 1 ft upstream of a bridge in the Rio Grande.



Non-uniform sediment distribution on the channel bed can be view when the channel flow is simulated. The cross section plot below displays the final cross section elevations in red compared to the cross section elevations at the start of the flood simulation shown in black. This image can be expanded to full view.



The user has an option in the water surface dialog box of plotting surveyed water surface and bed elevations along with the predicted values. To plot the surveyed water surface or channel bed elevation, the WSURF.DAT file must be created in the following format:

Optional WSURF.DAT file format:

Please note that PROFILES also has options for editing the channel bed slope and thalweg flow depth and for interpolating the slope and cross section geometry for

the cross section option. Refer the section on Pre-Processor Programs for a discussion on these features.

WSURF.DAT File Descriptors

2045	Line 1: # of channel elements with a surveyed ws elev.
4 4152.22	Line 2: Grid Element WS elevation
8 4151.84	Line 2: Grid Element WS elevation
...	

Notes:
Line 2: This line is repeated for each channel element with a surveyed ws elevation.

WSURF.DAT File Example

```
2045
   4   4152.22
   8   4151.84
  12   4151.69
  15   4151.55
  19   4151.41
  ....
```

The surveyed water surface can also be compared directly with the FLO-2D computed water surface in the WSTIME.OUT (see file description in the output file section) by creating a WSTIME.DAT file. The WSTIME.DAT file format is as follows:

The WSTIME.OUT file will contain:

Channel element number., time of survey (hrs), surveyed water surface elevation, computed water surface elevation, difference between the surveyed and computed water surface and cumulative difference between the surveyed and computed water surface elevations.

WSTIME.DAT File Descriptors

49			Line 1:	# of channel elements with a surveyed ws elev.
117632	4658.95	240	Line 2:	Grid Element WS elevation Time
117928	4655.80	240	Line 2:	Grid Element WS elevation
...				

Notes:
Line 2: This line is repeated for each data set.

WSTIME.DAT File Example

```
49
117632   4658.95   240
117928   4655.80   240
119882   4652.28   240
120580   4650.36   240
120915   4648.52   240
....
```

CHAPTER 7

DEBUGGING AND TROUBLE SHOOTING THE DATA FILES

7.1 BASIC TROUBLE SHOOTING GUIDELINES

Start simple and build on the flood model. After all the data files have been prepared, run a basic overland flood simulation. Review the results and if you have problems consult the troubleshooting tips found in this chapter.

The first step in debugging the data files after attempting a FLO-2D simulation is to review the ERROR.CHK file. As the model evolves the data error messages are more comprehensive and detailed. All the data errors recognized by the model are reported in this file. One of the most common errors is missing data that will invoke an end-of-file error statement to the screen. End-of-file messages are reported to the ERROR.CHK file. This error occurs when the model is searching for more data than is in the data file. Another common error is to activate a component or process switch without preparing the required data file. For example, an error will occur if MUD = 1 in the CONT.DAT file, but the data file SED.DAT is not prepared or if the sediment concentration in INFLOW.DAT is left out.

One data error that is difficult to locate is the array allocation violation where the array index number may be zero or larger than the assigned value. When this type of error is encountered, the FLO-2D model is terminated with an error message without indicating the file location or line entry of the error. To locate the data error, simplify the simulation and turn off all of the components and add them back to the model one at time until the error occurs again. You can reset SIMUL in CONT.DAT to the model time just after the error occurred to reduce time to debug the model. If your attempts to debug an error are ineffective, send

a zipped copy of the data files to FLO-2D (support@flo-2d.com) and briefly describe the problem.

You can create a set of backup data files to debug the model. Set IBACKUP = 1 in the CONT.DAT file. These backup files replicate your data files and will indicate if the computer is reading the data files correctly. The backup file should be identical to the original data file except for spacing. If the program terminates before reaching the first output interval timestep, there is probably an error in the data files. Start by checking the *.BAC files one by one. If one of the files is not complete, this may be the location of the data error.

Review the following files to analyze volume conservation problems: SUMMARY.OUT, CHVOLUME.OUT, CHANGRID.OUT, CHANMAX.OUT, TIME.OUT, BASE.OUT, ROUGH.OUT, CHANNEL.CHK, and SURFAREA.OUT. See the 'Pocket Guide' for further troubleshooting involving volume conservation, sticky grid elements listed in the TIME.OUT file, and numerical surging. The instructional comments at the end of each data file description in this manual contains a number of guidelines to assist the user in creating or checking the data files.

7.2 LIST OF COMMON DATA ERRORS

A list of the most common errors associated with running FLO-2D is presented below and a table for troubleshooting runtime errors follows the list. Whenever an error is encountered, refer to the ERROR.CHK file first.

TABLE 7.1. LIST OF THE THIRTEEN MOST COMMON DATA ERRORS	
1.	Missing data entries. Insufficient data was provided to the model.
2.	Switches were activated without the corresponding data or files (for example, see MUD, ISED, etc., in the CONT.DAT file).
3.	There was missing or additional lines in a data file when switch is activated. Observe the *** Notes: *** in the file descriptions.
4.	Percentages were expressed as a number instead of a decimal. See the description of XCONC in CONT.DAT or the HP(I,J,3) variable in INFLOW.DAT.
5.	The IDEPLT grid element was improperly assigned in INFLOW.DAT for the graphics mode.
6.	Channel infiltration switch INFCHAN was not 'turned on' in the INFIL.DAT file.
7.	Either one or both of channel and floodplain outflow elements were not assigned for a given grid element.

TABLE 7.1. LIST OF THE THIRTEEN MOST COMMON DATA ERRORS
8. Channel elements that are contiguous but do not share discharge were not identified with the NOFLOC variables in CHAN.DAT file.
9. The street width exceeded the grid element width.
10. The array size limitation for a variable was exceeded.
11. The available floodplain surface area was exceeded by assigning channels, streets, ARF's and/or multiple channels with too much surface area. Review the SURFAREA.OUT.
12. The rainfall variable R_DISTRIB data was entered as total cumulative rainfall instead of the percentage of the total rainfall (range 0.0 to 1.0).
13. The ISEDN switch for channel sediment transport was not 'turned on' in the CHAN.DAT file for the channel segment.

7.3 RUNTIME ERRORS

The following guidelines are provided to assist the user to resolve runtime errors. If the program stops executing after several output time intervals but has not reached the prescribed simulation time, the problem grid element(s) may be listed at the end of the BASE.OUT file. It may also be listed in the TIME.OUT file and it may appear in a dialog box that appears on the screen when the simulation terminates. The user should review the following data before revising the stability criteria:

- Review the problematic grid element elevation and Manning's n-value for the listed grid element as well as the contiguous 8 grid elements. This can be done in the GDS or FLOENVIR program.
- Check the channel data to determine if the grid element contains a channel and then check the relationship between the channel bed and bank elevations and the floodplain elevation within the grid element.
- Review the floodplain area available within the grid element including loss of floodplain area due to channels, streets, ARF's or multiple channels in the SURFAREA.OUT file.

Most volume conservation and numerical stability problems are associated with channel flow. When constructing a channel system, it is often necessary to fabricate cross section geometry, estimate roughness or adjust channel bed slopes. Mismatched channel morphology parameters are the primary source of numerical stability problems. To compute smoother hydraulics between two channel grid elements, adjust the bed slope, cross section flow area or roughness values. Try to avoid abrupt changes in cross sections geometry from one channel element to an-

other. The channel flow area should make a gradual transition from a wide, shallow cross section to a narrow deep cross section. An actual cross section transition may occur over several channel grid elements. Adjust the channel geometry so that the maximum change in flow area between channel elements is about 25%. To address channel problems, consider the following measures:

- Increase the roughness in wide, shallow cross sections and decrease the roughness in narrow deep channel grid elements.
- Reduce the difference between the cross section areas. Avoid abrupt cross section transitions between channel elements. Adjust the channel cross section geometry in the PROFILES. Using PROFILES, you can interpolate between surveyed cross sections.
- Review and adjust the bed slope with the PROFILES program.
- Select a longer channel length within the channel grid element.

7.4 TROUBLESHOOTING GUIDE

In addition to the following troubleshooting guide, refer to the 'Getting Started Guidelines' at the begin of this manual and the Pocket Guide to assist you in debugging runtime errors.

Program will not run

- Data errors. Turn off the component switches until the model runs.
- The executable program was damaged. Reload the program or contact technical support.
- The model is not properly licensed. Contact technical support.

Program stops

The model run is terminated before the first timestep or after a few timesteps with data file error indicated on the screen

- Review the ERROR.CHK file or the data file identified by the program error message.
- Review the backup file (*. BAC).
- Review the List of Common Data Errors.

Program stops

The model run is terminated after several timesteps indicating a numerical stability error. The grid element causing the stability error is listed on the screen instability dialog box or at the end of the BASE.OUT file.

Stability criteria were not met.

Review and revise the elevation and roughness data for the indicated grid element. The ROUGH.OUT and TIME.OUT files will help to locate the problem grid element. Also, check the contiguous grid elements to the problem element in the 8 directions.

Volume conservation

The volume conservation may indicate either a loss or gain of volume. A review of the SUMMARY.OUT and CHVOLUME.OUT will reveal if the volume conservation error is in the channel or on the floodplain. Volume conservation problems are indication of data error.

Discharge surging

Surging (alternating from low to high discharges) is usually associated with channel flow. Floodplain surging can occur but is less common. Review the maximum floodplain velocities in MAXPLOT and in the VELTIMEC. OUT and VELTIMEFP.OUT files. Unreasonable maximum velocities should be addressed. Other files to review for indications numerical surging include CHANMAX.OUT, HYCHAN.OUT, CHANSTABILTY. OUT, TIME.OUT, and ROUGH.OUT files. The hydrograph plots in the HYDROG program may display spikes to indicate surging. Some quick checks for data errors would include NOFLOC and NOEXHANGE assignments, missing OUTFLOW nodes, and overbank flow around an inflow node.

NOTE: Channel surging may occur and still meet volume conservation.

Supercritical flow

Supercritical flow is not necessary a problem, but its occurrence should be limited to conditions where it is expected such as in streets, concrete channels or steep bedrock watersheds. Supercritical flow on alluvial surfaces should be avoided.

Numerical Instability:

The channel surging may be related to numerical instability, abrupt changes in channel geometry, inappropriate slopes, supercritical flow or variable

mudflow sediment concentrations. Mismatched slope, flow area and n-values are the most common causes of channel instability. A combination of methods can fix instability.

- Abrupt changes in slope or severe adverse slope may cause instability. Use the PROFILES program to fix irregular bed slope conditions.
- Review the cross section flow areas over several channel elements in PROFILES. Eliminate any abrupt changes in cross section areas between channel elements. If the surging occurs at low flows, review only the bottom portion of the cross section not the bankfull conditions.
- Decrease the WAVEMAX and DEPTOL variables in the TOLER.DAT file. Suggested minimum values: WAVEMAX = 0.1; DEPTOLCH = 0.10.
- Insufficient floodplain area. Small floodplain surface areas can exacerbate unsteady flow. Review SURFAREA.OUT and increase the available grid element surface area for flood storage.
- Increase the n-values for the grid elements in the vicinity of the surging flow.
- Adjust the floodplain grid element elevations around the problem element.
- Increase the channel length within the grid element.
- The hydraulic structure discharge rating curve or table may be poorly matched with the upstream or downstream channel hydraulics. Review the hydraulic structure rating curve or table and compare the discharge values to those found in the HYCHAN.OUT file for that particular channel element.

Unexpected supercritical flow on alluvial surfaces:

- Adjust the limiting Froude number using the FROUDL variable in the CONT.DAT file or the FROUDC variable in the CHAN.DAT.
- Increase the floodplain or channel roughness values.
- Modify the slope. The grid elevations assigned by the GDS may not be representative of the field condition. Change the grid element elevations to make the channel or floodplain slope more uniform.

Variable mudflow sediment concentration:

- Review the sediment concentration in the inflow hydrographs in the INFLOW.DAT file.

FLO-2D simulation runs slow

Review the TIME.OUT file to identify the elements that have caused most of the timestep reductions. Small timesteps are the result of the model continually exceeding the numerical stability criteria for a small group of grid elements. The change in flow depth for a timestep may be too large. One of primary reasons for a slow flood simulation is that the relationship between the discharge flux and grid element surface area is poor. The rate of change in the discharge is too high for the selected grid element size. Increasing the grid element size is the best way to fix a very slow model. Other solutions may include:

- Adjust the channel geometry in transition reaches.
- Create a more uniform channel or floodplain slope.
- Revise the roughness values or limit the supercritical flow.
- Reduce the channel width, street width, ARF values or other parameters to increase the floodplain surface area. Review the SURFAREA.OUT file.
- Increase the grid element size (a last resort).

Graphics display is not activated

The graphics display is controlled by four variables: LGPLOT and GRAPTIM found in the CONT.DAT file and IDEPLT in INFLOW.DAT. IDEPLT is the floodplain or channel inflow grid elements whose hydrograph will be plotted on the display.

- Set LGPLOT = 2.
- Check the GRAPTIM variable for an acceptable update interval.
- Make sure that the IDEPLT variable matches an inflow grid element has a hydrograph assigned to it in INFLOW.DAT.

The inflow hydrograph does not plot in the graphics display

- No hydrograph is associated with the IDEPLT variable.
- The hydrograph duration is too long. Reduce the hydrograph length.
- The rainfall duration is too long. Reduce the rainfall time.
- Inappropriate peak discharge or total rainfall values distort the scale for hydrograph plot.

Program stops. Flow depths exceed 100 ft

Flow depths greater than 100 ft cause the FLO-2D program to terminate. If flow depths greater than 100 ft are expected consult technical support to have the model code revised. If depths of 100 ft are not expected, then excessive ponding or surging is occurring.

- Identify the problem element in MAXPLOT or check the end of the BASE.OUT file to identify the problem grid.
- Check TIME.OUT to determine if the problem element is also causing the model to run slowly.
- Check the elevation of the problem grid element in the FPLAIN.DAT or in the GDS.
- If the depressed element is a gravel pit or some other real feature, increase the n-value to decrease the vertical velocity into the pit.

Erratic discharge in the channel elements.

A review of plotted hydrographs in HYDROG or an examination of the CHANMAX.OUT or HYCHAN.OUT files will reveal if the flow discharge between contiguous channel elements is surging with spikes when a consistent rise or fall of the downstream discharge is expected.

Channel surging can be natural phenomena. Rivers rise and fall over a few tenths of a foot in matter of seconds. During high flow, the variation in discharge associated with stage change ~0.2 ft can be 1,000 cfs or more. If no channel surging is expected, there are several solutions:

- Channel grid elements that are contiguous but do not share discharge must be identified in the CHAN.DAT file. NOFLOC variables in the CHAN.DAT may not be properly assigned. See the CONFLUENCE.OUT file. The NOFLO1 or NOFLO2 grid elements were misidentified.
- The channel grid elements in the CHAN.DAT file (ICHANGRID) in line 3 are mislabeled.

Erratic flow in the floodplain grid elements.

Erratic flow in the floodplain grid elements is usually the result of errors in the FPLAIN.DAT file. This type of error generally occurs when the user edits the FPLAIN.DAT file manually and adds, subtracts or moves grid elements around. Virtually all erratic flow conditions on the floodplain can be corrected by revisions either to n-values or elevations in the GDS or FLOENVIR.

Channel extends through another channel element.

The right channel bank assignments are automated in the GDS and FLOENVIR. The multiple left bank elements can be assigned to the same right bank on a river bend. The line connecting the left and right bank elements cross other bank connecting lines the model will result in an error message reported in ERROR.CHK file.

The channel bank elements can be viewed in the GDS and FLOENVIR. If there is a problem with the channel outline, simply revised the right bank element. The right bank element can be any grid element if it does not cross another connecting channel bank line. A right bank element can connect to more than one left bank element.

Program stops; identifying one or more grid elements with too little floodplain surface.

The model will report an error in ERROR.CHK if the channel right bank has is too little surface storage area on the floodplain portion of the element. If this problem occurs and the floodplain surface is less than 5%, then there are several solutions:

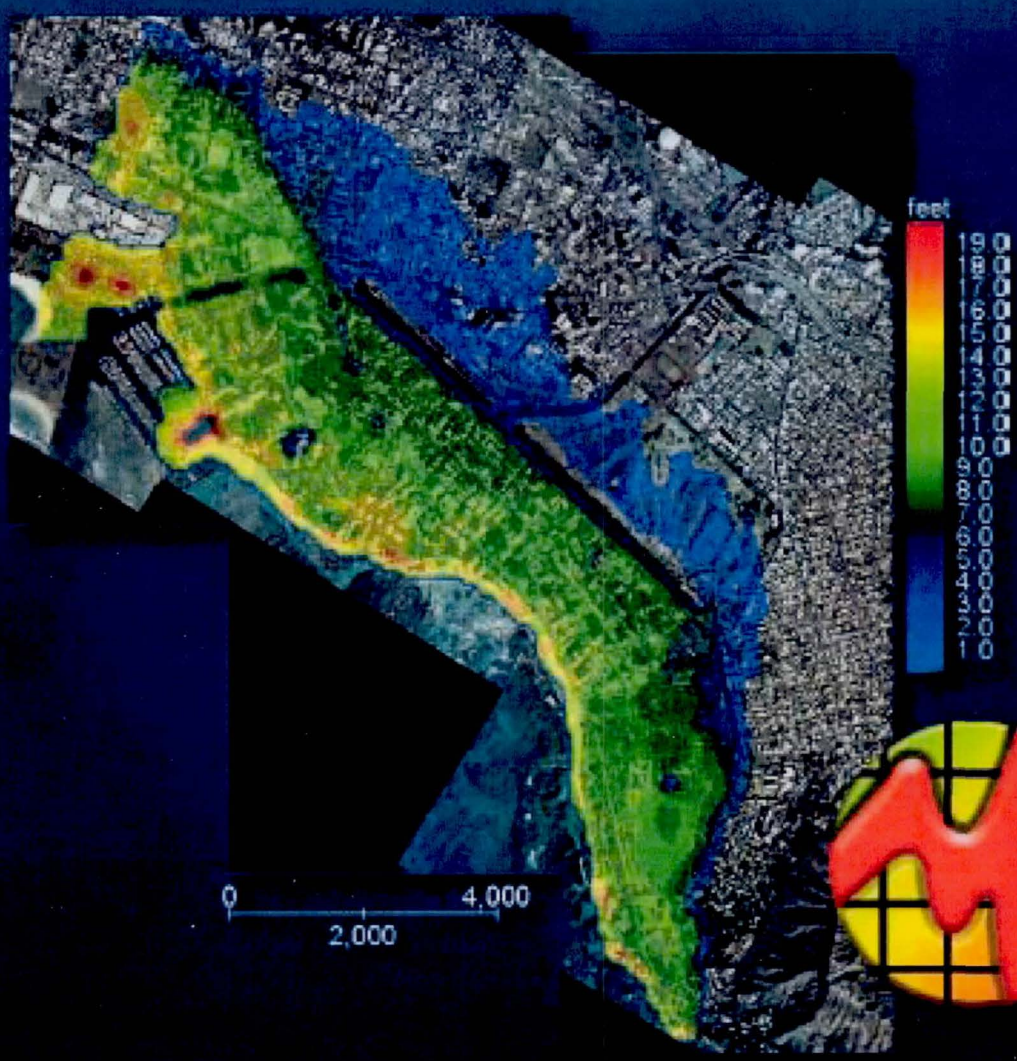
- Reduce the ARF value, multiple channel area or street area.
- The channel area can be reduced by decreasing the XLEN variable or top width, which is a function of the channel in the natural channels, the side slopes, or the bottom width in the trapezoidal cross section or the width in the rectangular cross section.
- As a last resort the grid element size can be increased, but this requires the re-generation of the grid system.

CADPTS.DAT error

If grid elements are added or deleted to grid system and the FPLAIN.DAT file is revised, the CADPTS.DAT will report errors if it is not also modified. The CADPTS.DAT file must be changed to reflect the revisions in the FPLAIN.DAT file. Use the CHECKER program to verify the accuracy of the revisions in the FPLAIN.DAT file.

FLO-2D

MAPPER MANUAL



Version 2009

Table of Contents

Page

I. Introduction 1

1.1 Getting Started with *Mapper* 2

1.2 Display the Initial Flow Depth Plots 4

II. Toolbar Plotting Buttons 8

2.1 Plot of grid element ground surface elevation 9

2.2 Plot of grid element maximum water surface elevation 9

2.3 Plot of grid element maximum flow depth 9

2.4 Plot of grid element velocities 10

2.5 Impact force and other variable plots 10

2.6 Time for Depth 11

2.7 Plot of grid element maximum deposition, maximum scour and final bed elevation 11

2.8 Plot of DTM point ground surface elevation 11

2.9 Plots of DTM point maximum flow depth 11

III. Mouse Button Commands 12

3.1 Zoom in (left mouse button) 12

3.2 Color scale repositioning (left mouse button) 12

3.3 Distance scale repositioning (left mouse button) 12

3.4 Right Click on Contour Graphs 13

IV. Legend and Scale Controls 14

V. Hazard Maps 15

VI. Flood Inundation Contour Maps 20

VII. Example Plots 24

VIII. Menus and Toolbar 29

8.1 File Menu Commands 29

Read FLO-2D Results Command… (*File Menu*) 29

Import Image/Individual Image (File Menu) 30

Import Image/Image Catalog (File Menu) 31

Import Elevation Points Command (File Menu) 33

Import DTM Points…Command (File Menu) 33

Individual ArcInfo ASCII Grid File Command (File Menu) 34

ArcInfo ASCII Grid File Catalogue (File Menu) 35

Import ASCII Grid File Command (File Menu) 37

Save Elevation Point Command (File Menu) 38

Create Shape File for Current Tim/Flow Depths (File Menu) 39

Create Shape File for Current Tim/Flow Depth (Shaded Contours) (File Menu) 40

Create Shape File for Current Tim/Velocities (File Menu) 40

Create Grid Shape File (File Menu) 41

Export…Command (File Menu) 42

Copy... Command (File Menu) 43
Print... Command (File Menu) 44
Create FLO2DGIS.OUT... Command (File Menu) 44
Exit (File Menu) 45
8.2 View Menu Commands 46
100% Command (View Menu) 46
Layers List Command (View Menu) 48
Grid Element Numbers Command (View Menu) 50
Display Grid System Command (View Menu) 51
Display Boundary Cells Command (View Menu) 52
Display Title Command (View Menu) 53
Display Exclusion Polygons Command (View Menu) 54
Display Legend and Scale (View Menu) 54
Display All Cells Groups (View Menu) 54
Display All Slices (View Menu) 55
Display Area of Inundation (View Menu) 56
Track Elevation Points Command (View Menu) 56
Table of Variables (View Menu) 57
Highlight Grid Element Number... (View Menu) 58
Redraw Command (View Menu) 58
8.3 Tools Menu Commands 59
Option Directory Paths Command (Tools Menu) 59
Interpolate to DTM Points Command (Tools Menu) 60
Create Elevation Points Layer Command (Tools Menu) 61
Delete Elevations From Selected Area Command (Tools Menu) 62
Create Grid Layer Command (Tools Menu) 62
Create Area of Inundation Layer Command (Tools Menu) 64
Exclusion Polygons Command (Tools Menu) 64
Plot Flow Depth Profile Command (Tools Menu) 67
Map/View Map Command (Tools Menu) 69
Map/View Text Handles Command (Tools Menu) 71
Flooding Damage/Estimate Flood Damage Command (Tools Menu) 71
Time Plots Command (Tools Menu) 77
Hide All Plots (Time Plots Shortcut Menu): 79
Delete Cell Group (Time Plots Shortcut Menu): 79
Show Tree of Graphs (Tools Menu): 79
8.4 Help Menu Commands: 80
Contents Command (Help Menu) 80
IX. References 81

I. Introduction

FLO-2D Mapper is a post-processor program that creates maps and other plots of the *FLO-2D* model results including hydraulic variables, water surface elevations, duration of inundation, impact force, static pressure, specific energy, sediment scour or deposition and others. Three types of map plots can be generated:

- Grid element plots where each element is assigned a color depending on the value of the selected plot variable.
- Line and shaded contour maps based on the grid element values.
- DTM points to generate detailed flow depth contour plots based on grid element water surface elevations and DTM point ground elevations.

Mapper can also generate depth and velocity versus time graphs at user selected locations, flow depth profiles along user defined sections, flood damage plots, and hazard maps. *Mapper* utilizes the capabilities of the MS-Windows™ operating system. The plots can be constructed interactively using mouse point and click events. *Mapper* supports the English and the SI system of units. *Mapper* includes the following features:

- ESRI MapObjects© GIS software controls.
- ESRI shape file formatted data for land use, soil types, and Manning roughness coefficients, etc. can be imported.
- ESRI ArcInfo ASCII grid files containing terrain elevations can be imported.
- Multiple geo-referenced aerial photos in various graphic formats can be imported such as TIFF, BMP, JPG, etc. as background to flooding results.
- Multiple ESRI ArcInfo ASCII grid files listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- Multiple image files such as aerial photos listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- Multiple layer edit capability, including control of layer properties.
- Zoom features and a pan display enable views to be adjusted.
- Polygon and mouse painting can be used to selected grid elements.
- DTM points can be deleted and DTM point files saved.
- A map template for supporting map creation and editing is available.
- Grid element, line and shaded contour plots of the following FLO-2D results:
  - ✓ Ground surface elevation
  - ✓ Maximum water surface elevation
  - ✓ Floodplain maximum and final flow depth
  - ✓ Floodplain maximum and final velocity
  - ✓ Channel maximum flow depth
  - ✓ Combined maximum channel and floodplain flow depth
  - ✓ Combined maximum channel and floodplain flow velocity
  - ✓ Street flow maximum depth and velocity
  - ✓ Maximum sediment scour and deposition
  - ✓ Final bed elevation
  - ✓ Duration of inundation
  - ✓ Impact force
  - ✓ Static pressure

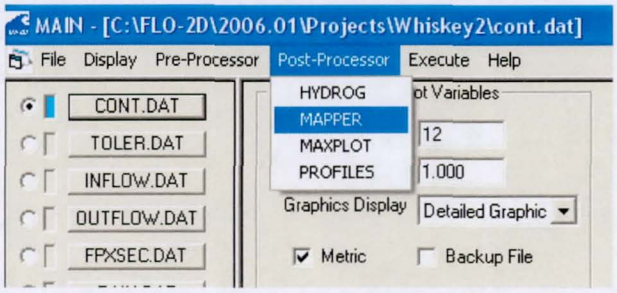
- ✓ Specific energy
- ✓ Levee freeboard deficiency
- ✓ Time of flood to 1ft, 2ft, and maximum depth.

- Velocity vectors can be displayed corresponding to the velocity magnitude.
- Flood animation can be viewed to review floodwave progress.
- Plot of depths versus time or velocity versus time at any user selected grid element or group of grid elements.
- Flow depths and topography profile cuts can be plotted.
- Flood damage ($$) can be estimated based on cost tables for inundated buildings.
- Hazard maps based on user defined depth and velocity times depth thresholds as function of flood frequency can be generated.
- DTM shaded contour plotting algorithm based on Triangulated Irregular Network (TIN) generation.
- User defined exclusion areas from the shaded contour plots.
- Automatically generated ESRI shape files for all map plots that can be imported into ArcGIS® for further processing.
- Inline help system.

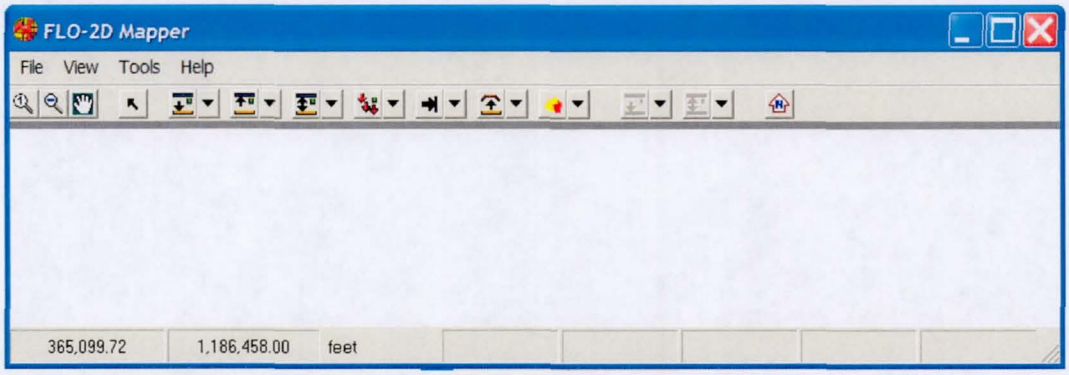
IMPORTANT NOTE: *For the Mapper system to operate properly, the MS-Windows ™ environment must be set to a number format with decimal point using the International (Regional) option of the Control Panel.*

1.1 Getting Started with *Mapper*

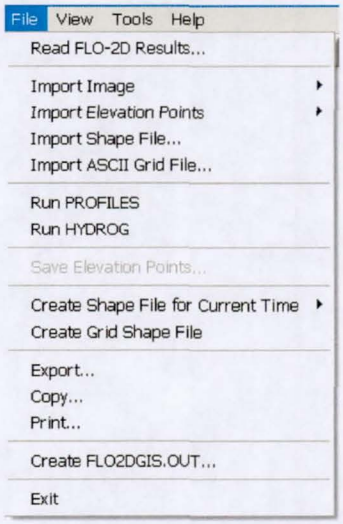
Mapper is automatically installed with the FLO-2D model installation. The *Mapper* program is located in the FLO-2D subdirectory and an icon can be placed on the desktop. To start the program by double-clicking the *Mapper* icon or on the *MAPPER.EXE* file in Explorer. You can also start by clicking on *Mapper* in the GUI Post-Processor pull-down menu:



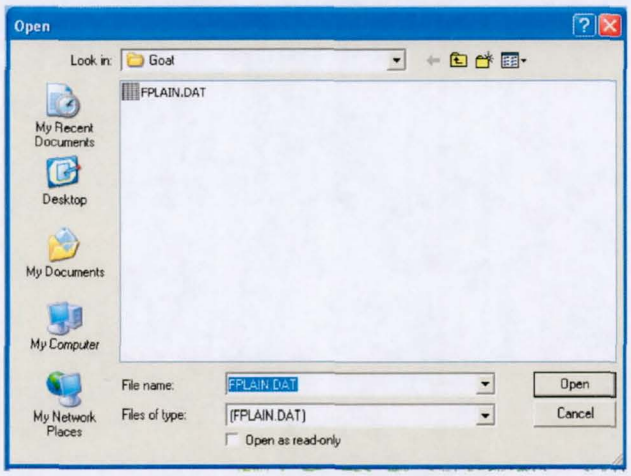
Mapper main window will appear:



To load FLO-2D results use the *Read FLO-2D Results* Command… in the File Menu.

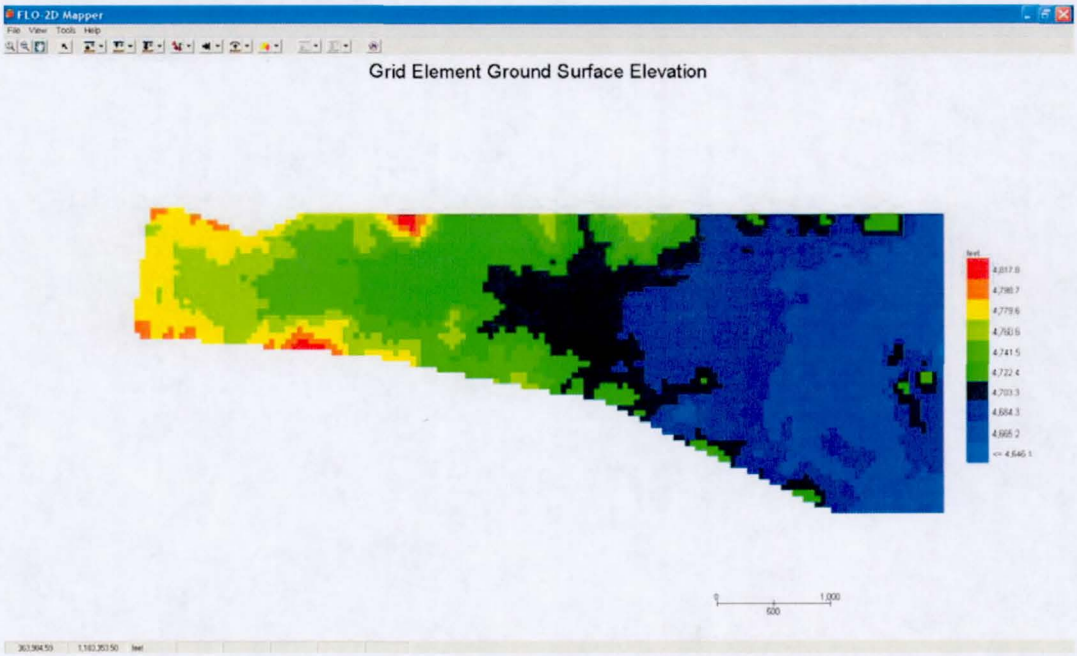


A dialog box appears to select the project FPLAIN.DAT file:



You need only to select the FPLAIN.DAT data file. The other required and companion files are automatically read from the same directory. Once the files are loaded, *Mapper* will display the

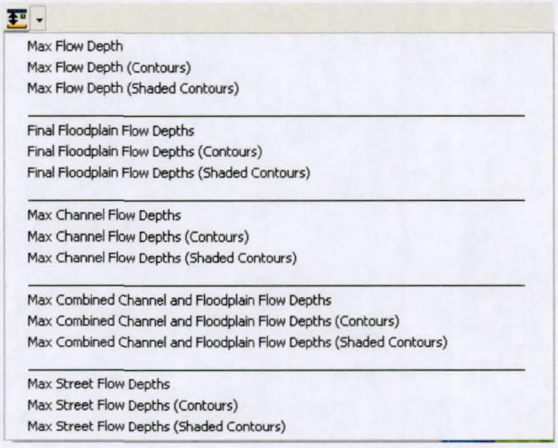
Ground Surface Elevation density plot:



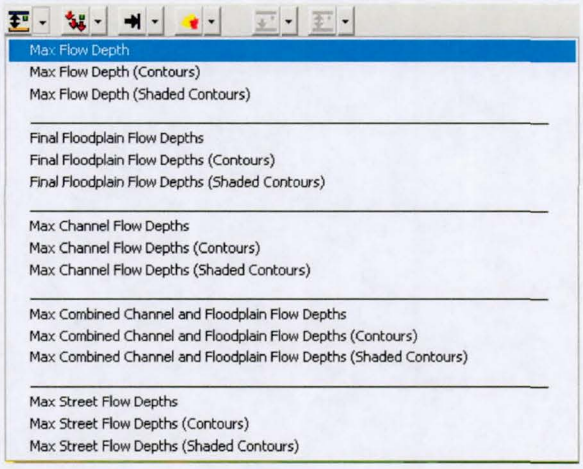
IMPORTANT NOTE: *You must have completed a FLO-2D flood simulation and the *.OUT files have to exist in the same directory as the FPLAIN.DAT file.*

1.2 Display the Initial Flow Depth Plots

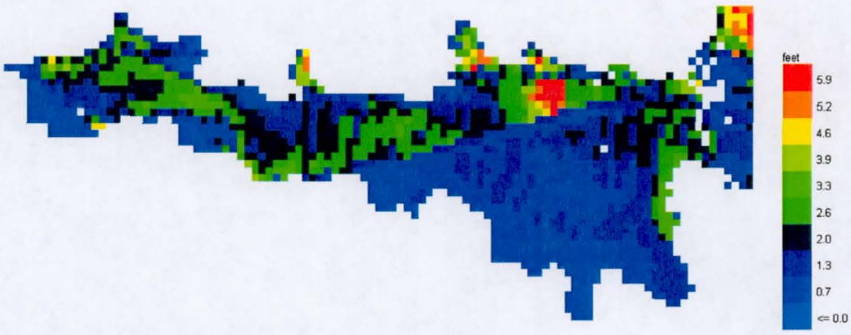
The initial flow depth maps that *Mapper* creates are the grid element plots where each grid element is assigned a color depending on the value of the selected plot variable and lines and shaded contours plots. To create a plot of grid element maximum depths, click drop-down arrow besides the Depth Plot Icon . You will get the following drop-down menu:



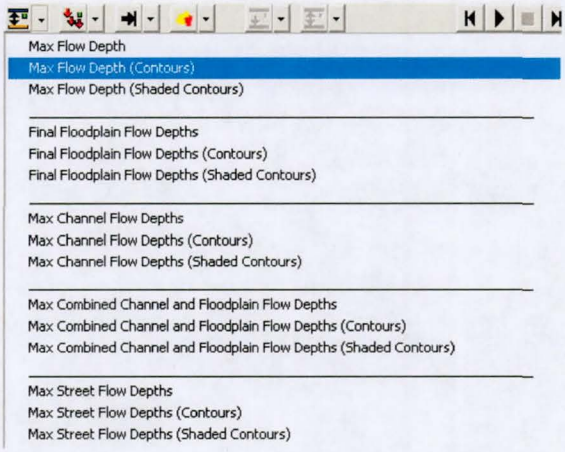
Each plot group has three options: a grid element plot, a line contour plot and a shaded contour plot. To display the grid element maximum flow depth, click *Max Flow Depth*:



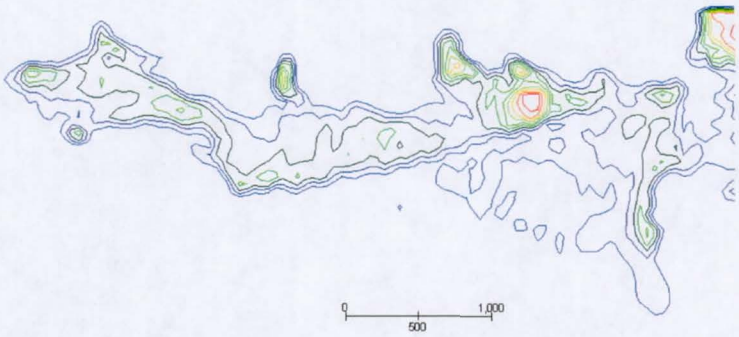
An example of the *Mapper* maximum depth plot:



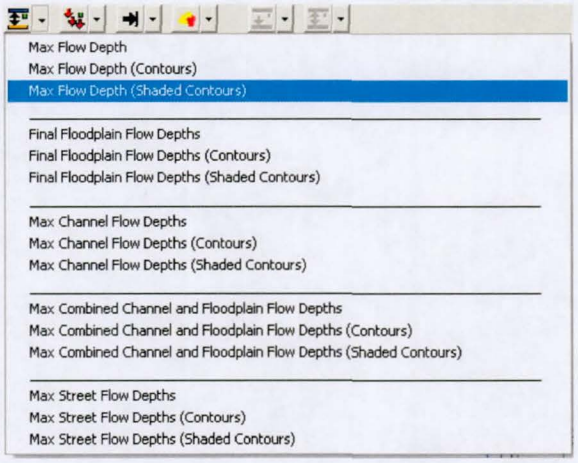
For the maximum depth line contour plot, click on *Max Flow Depth (Contours):*



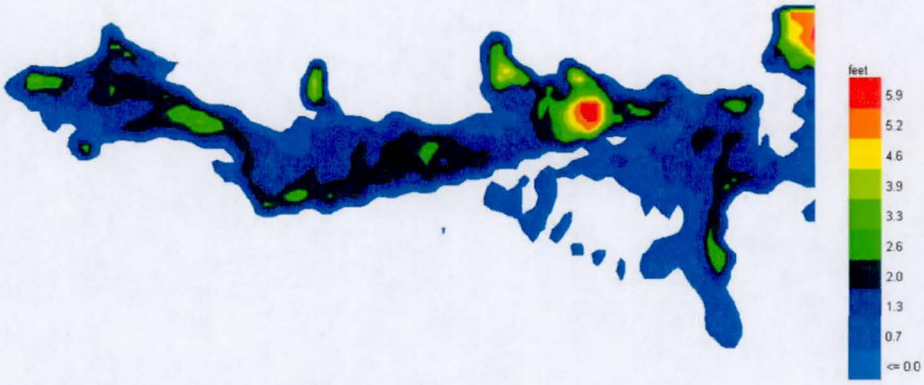
Mapper will display the maximum flow depth contour lines plot:



Finally, to display the shaded contours for the maximum depth, click on *Max Flow Depth (Shaded Contours):*



The maximum depth shaded contour plot in *Mapper* is displayed as follows:



Mapper plots for most of the other FLO-2D output variables are constructed in a similar manner.

II. Toolbar Plotting Buttons

Most *Mapper* plots are accessible through the toolbar:

The toolbar contains buttons that generates maps of the *FLO-2D* results. Note that some or the toolbar buttons are grayed out because they are unavailable if *Mapper* does not find the required FLO-2D output files. Each icon button has a drop-down arrow to access the available plots.

Grid element based plots

Ground Surface Elevation

Maximum Water Surface Elevation

Maximum Flow Depth (Floodplain)
Final Floodplain Flow Depth
Maximum Channel Flow Depth
Maximum Combined Channel and Floodplain Flow Depth
Maximum Street Flow Depth

Maximum Velocity (Floodplain)
Maximum Street
Maximum Combined Channel and Floodplain Flow Velocity
Final Flow Velocity

Duration of Inundation
Impact Force
Static Pressure
Specific Energy
Levee Freeboard Deficiency

Time for Depth

Hazard Maps

Maximum Deposition
Max Scour
Final bed elevation

DTM based plots

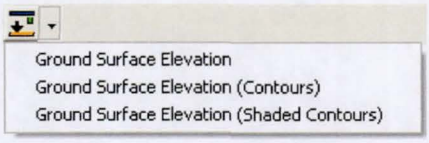
Ground Surface Elevation

Maximum Flow Depth

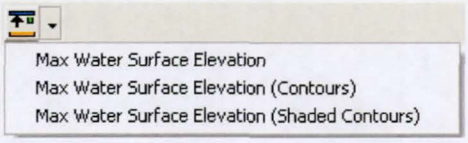
Digital Flood Insurance Rate Map Tool (explained in a separate manual)

Every icon has a drop-down arrow that displays a menu for each available plot as follows:

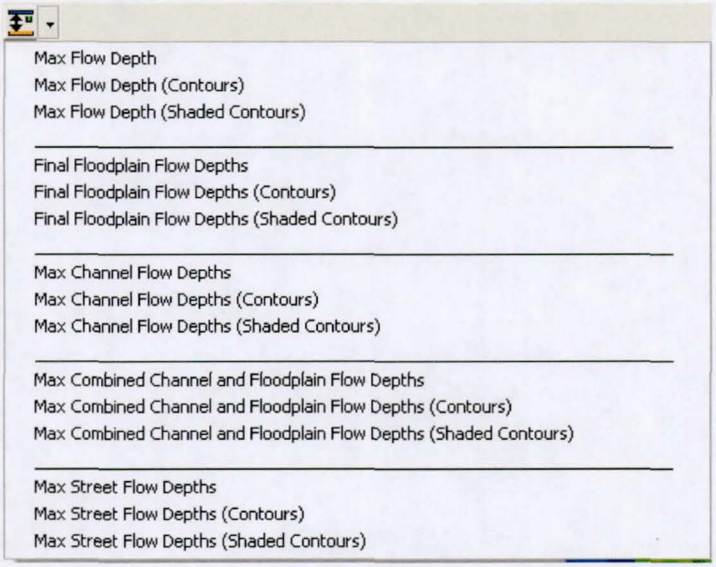
2.1 Plot of grid element ground surface elevation



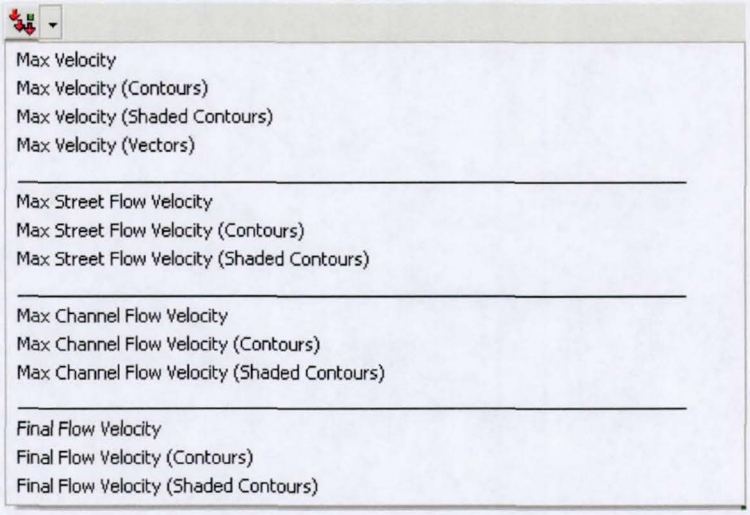
2.2 Plot of grid element maximum water surface elevation



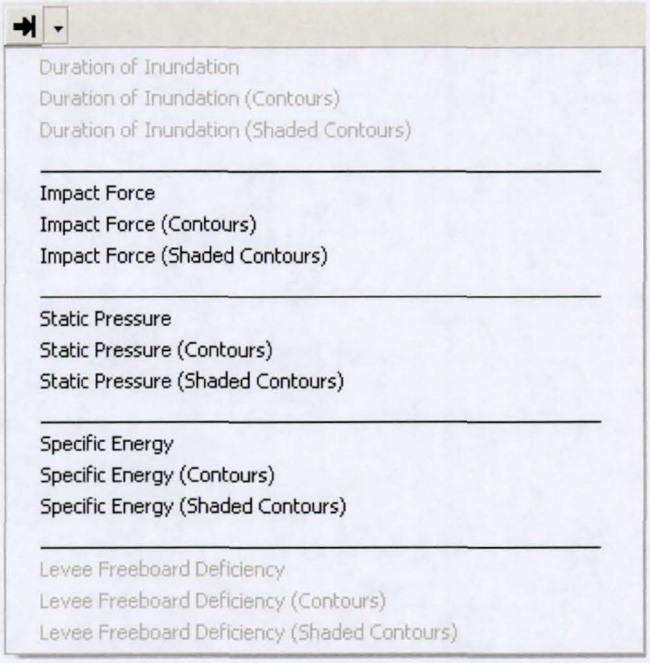
2.3 Plot of grid element maximum flow depth



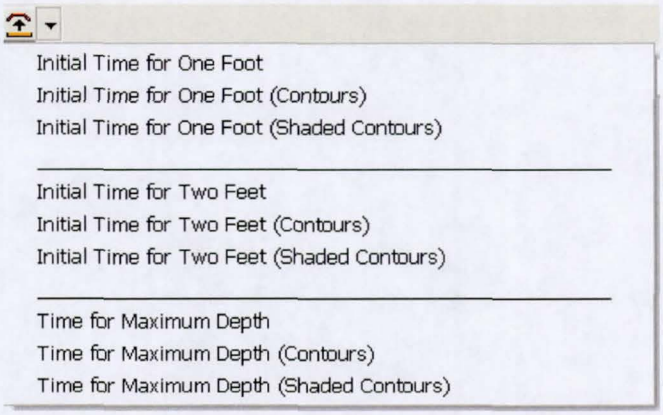
2.4 Plot of grid element velocities



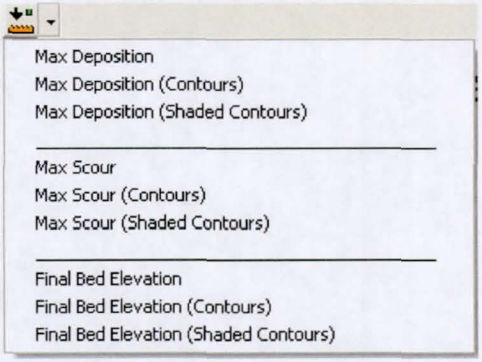
2.5 Impact force and other variable plots



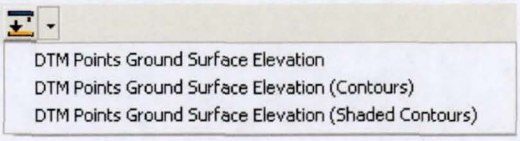
2.6 Time for Depth



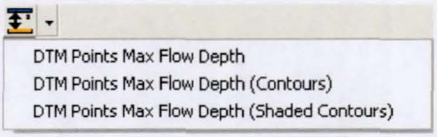
2.7 Plot of grid element maximum deposition, maximum scour and final bed elevation



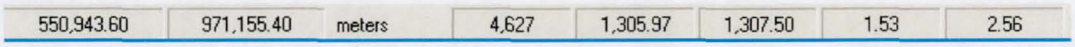
2.8 Plot of DTM point ground surface elevation



2.9 Plots of DTM point maximum flow depth



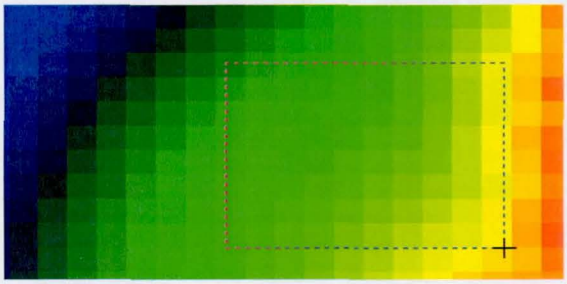
The left side of the bottom tool bar also has data boxes that display the x-coordinate, y-coordinate, grid element number, ground and water surface elevation, and maximum flow depth and velocity when the mouse cursor passes over the grid element point on the flow domain.



III. Mouse Button Commands

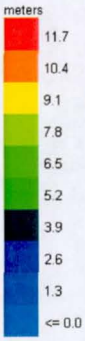
3.1 Zoom in (left mouse button)

You can zoom-in on the working region using a rectangular viewing area. Click the left mouse button to position the pointer on one the vertices of the desired area and drag the mouse to the opposite vertex. The mouse pointer changes to a cross-shaped pointer and a rectangle appears to outline the selected area. When the left button is released, the selected rectangular area is magnified to the full screen.



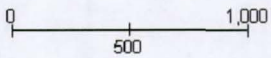
3.2 Color scale repositioning (left mouse button)

A color scale is usually placed at the right of the graphs. The user can reposition it by left clicking the mouse on the scale and dragging it to the desired position.



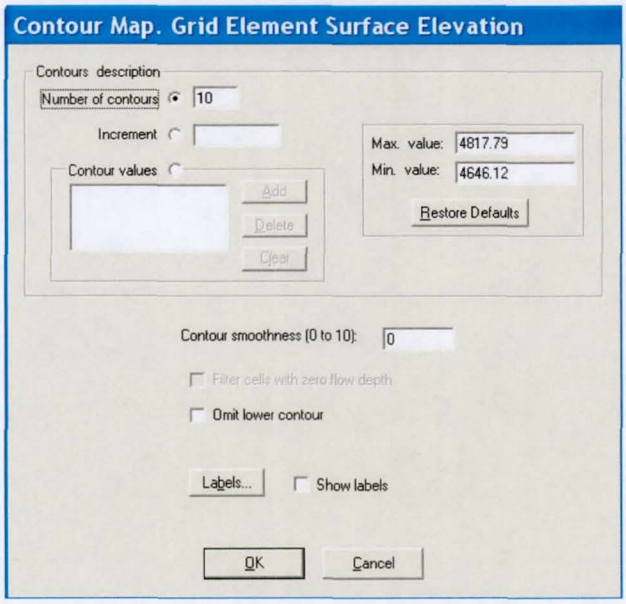
3.3 Distance scale repositioning (left mouse button)

A distance scale is placed at the right of the graph below the color scale. The user can reposition it by left clicking the mouse on the scale and dragging it to the desired position.



3.4 Right Click on Contour Graphs

This option is activated by clicking the right mouse button over the displayed plot. Contour intervals and other options can be adjusted for all line contour and shaded contour plots. After clicking the right mouse button on the plot, a dialog box appears:



In this dialog box, you can select between three options to display contours or shaded contours:

- Define the *Number of contours* to display equal increment contours between the minimum (*Min. value*) and maximum (*Max. value*).
- Specify a contour *Increment* to display the contours in successive increments starting with the minimum value (*Min. value*).
- Specify a list of *Contour values* you want to display. You may *Add* or *Delete* any value from the list and *Clear* the whole list.

The remaining options for all hydraulic variables include:

Contour smoothness (0 to 10) selects a level of resolution for the contours. A higher value for the contour smoothness requires more points for the contour interpolation, but it will also significantly increase the contour computation time.

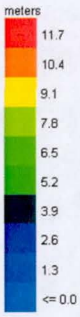
Omit lower contour eliminates the lower contour.

Gradual contour construction selects an alternative contour building algorithm that will produce better contour graphics for certain plots.

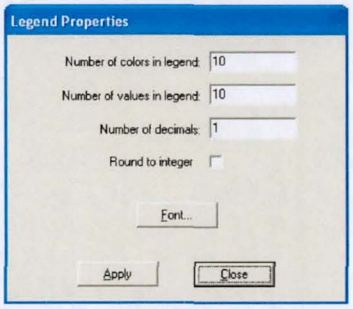
Filter cells with zero flow depth works only for flow depth contour plots. When this option is selected, only the grid elements with flow depth greater than zero are considered when developing the contours.

IV. Legend and Scale Controls

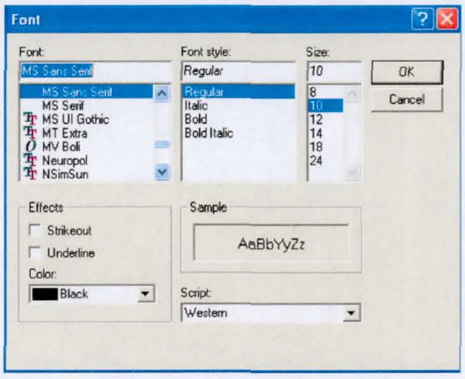
Mapper provides a number of functions to edit and customize the plot legend. Initially the legend is automatically set to the range of values of the selected plot. The legend can be dragged to any location on the plot. The following legend shows 10 ranges using the rainbow colors:



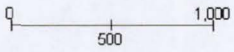
To customize the legend, right click on legend to display the following dialog box:



You may change the number of colors (range values that default to 10), the number of values (numbers associated with the color ranges), and the number of decimal places. You may also use check box to round the numbers to the nearest integer value. Using the *Font...* button, you can display the font editor dialog box to change legend font type, style and size:



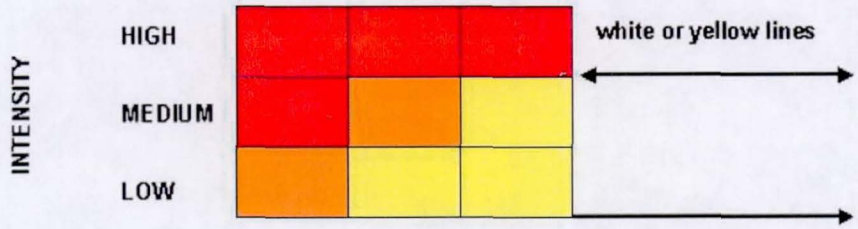
The scale indicates a reference distance corresponding to the plot scale. You can also drag the scale to any position in the plot window.



V. *Hazard Maps*

Mapper can create flood hazard maps based on a methodology developed by Garcia et al. (2003, 2005). The hazard delineation map criteria used in *Mapper* were first proposed in the PREVENE, 2001 project, where it was applied to two alluvial fans in Caracas, Venezuela and later adapted and applied to other urbanized alluvial fan Garcia et al. (2003, 2005). This approach follows Swiss and Austrian standards that establish three zones to delineation the flood hazard level (for more discussion refer to OFEE et al. 1997, Fiebiger 1997).

Flood hazard level at a specific location is 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. Large flood events occur less frequently. Flood hazard level is then defined as a discrete combined function of the event intensity (severity of the event) and return period (frequency) as shown in the following figure.



	HIGH	MEDIUM	LOW	VERY LOW
PROBABILITY				
Return Period	**10**	**100**	**500**	**>>500**
Probablity of Exceedence	**10%**	**1%**	**0.2%**	**<<0.2%**

High Hazard (red)

Medium Hazard (orange)

Low Hazard (yellow)

In the above figure, the probability limits are defined for return periods of 10, 100 and 500 years. The user can select other return periods to construct the flood hazard map. A flood hazard map is based on the three color levels to define high (red), medium (orange) and low (yellow) level. These map colors translate into specific potential hazard areas as shown in the following table.

FLOOD HAZARD DEFINITION		
Hazard Level	Map color	Description
High	Red	Persons are in danger both inside and outside their houses. Structures are in danger of being destroyed.
Medium	Orange	Persons are in danger outside their houses. Buildings may suffer damage and possible destruction depending on construction characteristics.
Low	Yellow	Danger to persons is low or non-existent. Buildings may suffer little damages, but flooding or sedimentation may affect structure interiors.

To define the event intensity, similar methods use a combination of flow depth and velocity. The Austrian method (Fiebiger et al. 1997) uses the total energy defined by:

$$h + v^2/2g$$

where h is the flow depth, v is the velocity and g is the gravitational acceleration. The Swiss method (OFEE et al. 1997) defines the intensity in terms of a combination of flow depth h and the product of h and v. This approach enables high intensity to be assigned to high flow depths independently of the flow velocities. Following the Swiss method, the flood intensity criteria used in *Mapper* make the distinction between water flooding and mud or debris flows. Intensities are defined in terms of the maximum water depth and the product of the maximum velocity multiplied by the maximum depth. These criteria to define flood intensity differ from the Swiss method in several aspects. For example, the Swiss method for debris flow intensity estimation establishes that a high intensity event occurs when $h > 1.0$ m AND $vh > 1.0$ m^2/s. Considering the specific conditions in project, it may necessary to change the hazard level thresholds. In *Mapper* the user can input values for flow depth and velocities that define the intensity thresholds. For the case of water flooding, the flood intensities could be defined by the selection of values in the following table.

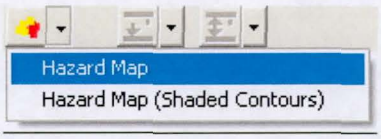
DEFINITION OF WATER FLOOD INTENSITY			
Flood Intensity	Maximum depth h (m)		Product of maximum depth h times maximum velocity v (m^2/s)
High	$h > 1.5$ m	OR	$v\,h > 1.5$ m^2/s
Medium	0.5 m $< h < 1.5$ m	OR	0.5 $m^2/s < v\,h < 1.5$ m^2/s
Low	0.1 m $< h < 0.5$ m	AND	0.1 $m^2/s < v\,h < 0.5$ m^2/s

Mud or debris flow events are more destructive than water floods, therefore, the mudflow intensity criteria are more conservative.

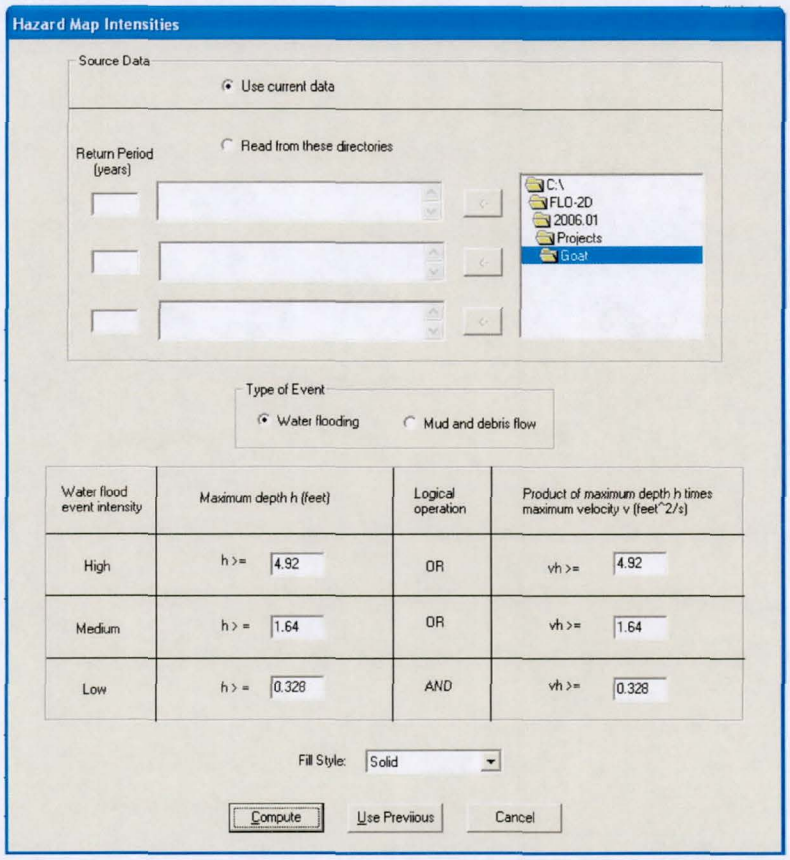
DEFINITION OF MUD OR DEBRIS FLOW INTENSITY			
Flood Intensity	Maximum depth h (m)		Product of maximum depth h times maximum velocity v (m^2/s)
High	$h > 1.0$ m	OR	$v\,h > 1.0$ m^2/s
Medium	0.2 m $< h < 1.0$ m	AND	0.2 m $< v\,h < 1.0$ m^2/s
Low	0.2 m $< h < 1.0$ m	AND	$v\,h < 0.2$ m^2/s

The hazard criteria encompass the probability of occurrence of a water or mudflow event for three selected returns periods. This requires the FLO-2D simulation of the three selected flood events. The model predicts the maximum depths and velocities for each return period flood. For each grid element, the event intensity for a return period flood determines the hazard based on the above criteria.

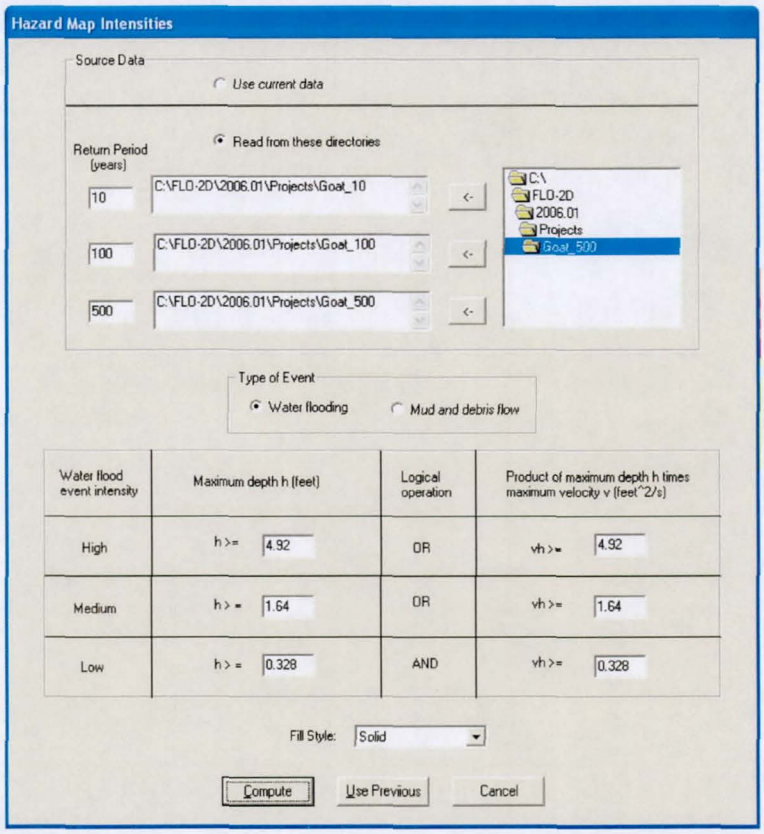
The Hazard mapping function is accessed by the button: . Clicking the drop-down list arrow and Hazard Map:



The following dialog box is displayed:



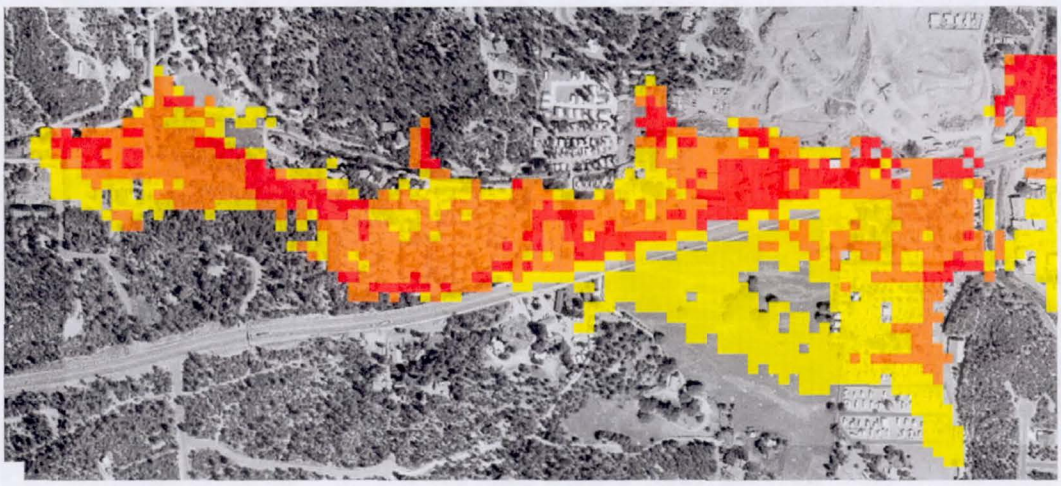
You have two principal options in the dialog box: 1) *Use current data* in the project folder; or 2) Read the FLO-2D results for three completed return period flood events. For the latter case you need to have completed flood simulations in separate directories as shown in the following dialog box.



For the above project, there are three flood simulations:

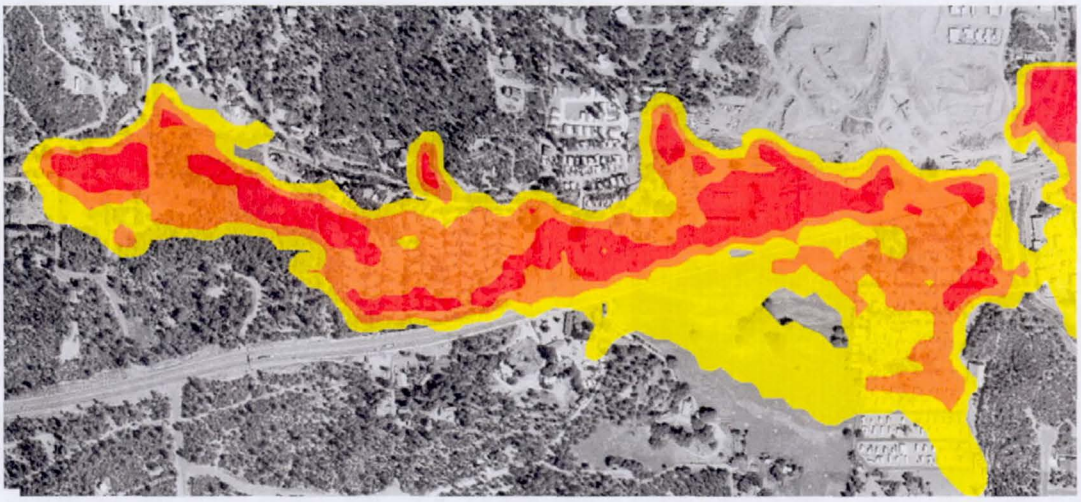
1) A 10-year return period flood in directory C:\FLO-2D\2006.01\Projects\Goat_10.
2) A 100-year return period flood in directory C:\FLO-2D\2006.01\Projects\Goat_100.
3) A 500-year return period flood in directory C:\FLO-2D\2006.01\Projects\Goat_500.

The resulting hazard map appears as follows:



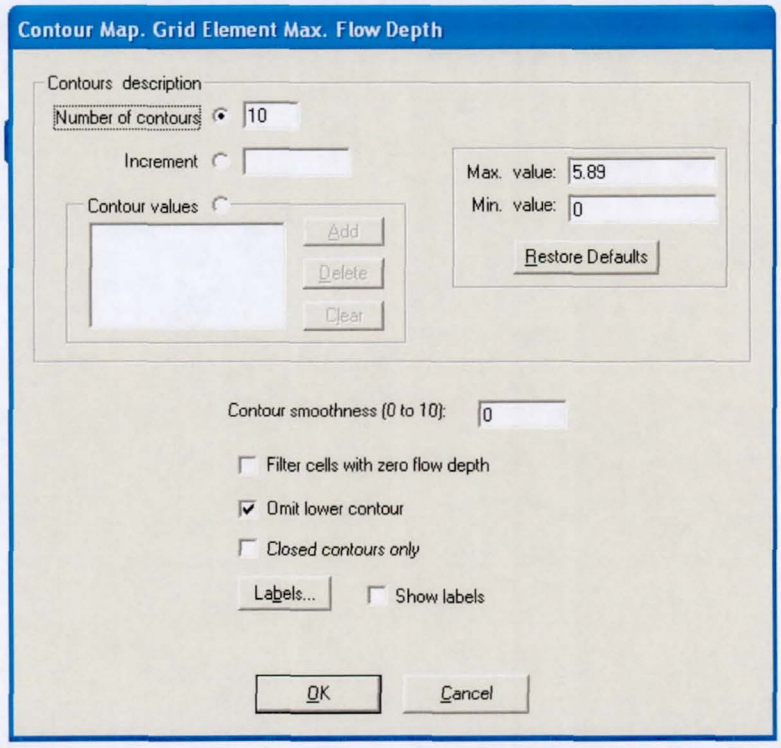
The colored grid elements represent the low, medium and high flood hazard. The corresponding

shaded contour plot is initiated from the pull-down menu:



VI. Flood Inundation Contour Maps

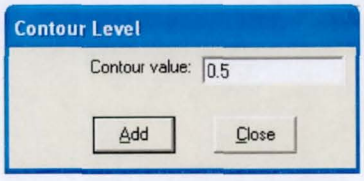
Mapper allows you to customize the line and shaded contour map plots. To initialize the editor dialog box, right-click anywhere on the contour map.



First, you may want to adjust the number of contours or the contour increment. There are three options:

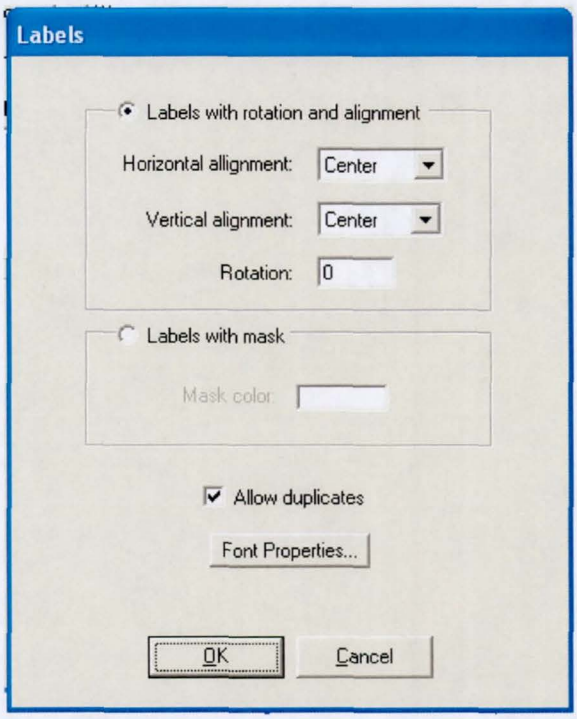
- *Number of Contours*: Edit the number of line or shaded contours. The contour value interval (increment) is equal to (Max. value /Min value)/ (Number of contours).
- *Contour Increment*: Selecting this option requires inputting the contour increment in the text box. The lines or shaded contours will start at the minimum value *(Min. value)* and continue to the contour closest to the maximum value *(Max. value)*.
- *Individual Contour Values*: Use this option to assign specific contour values.

To enter specific contour increment values, first click the *Contour values* radio button, and then click the Add button. The following input box will be displayed:



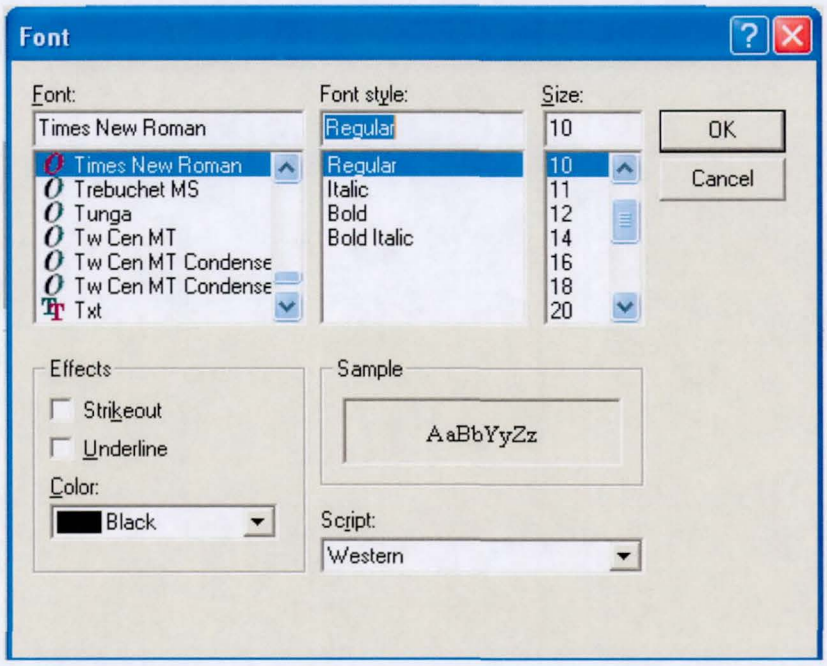
Input the desired contour interval value and click on the *Add* button. Repeat this procedure for all desired contour increment values.
The following definitions and controls are available in the above dialog box:

- *Max. value*: The maximum value for the variable being plotted. *Mapper* automatically determines the Max. value. You can adjust the maximum contour value by editing the value in the text box.
- *Min. value*: The minimum value for the variable being plotted. *Mapper* automatically determines the Min. value. You can change the minimum contour value by editing the value in the text box.
- *Restore Defaults button*: If you edit the *Max* and *Min values*, click this button to recover the original values in the FLO-2D variable output file.
- *Contour smoothness*: Select a value between 0 and 10 to smooth the contour lines and shaded contour outlines. A value of 10 is the maximum contour smoothing and takes the longest computational time. A smoothing factor of 3 is suggested for an initial adjustment.
- *Filter cells with zero flow depth* check box: Checking this box will omit all grid elements that may have zero values from the contour interpolation calculation.
- *Omit lower contour*: Checking this box will eliminate the lowest contour line or shaded contour from the plot.
- *Closed contours only*: Checking this box will result in plotting only closed contour lines or shaded contours that do not end at the grid system boundary.
- *Show labels check box*: Check this box to show the contour line labels.
- *Labels... button*: This button allows editing the contour lines labels. A *Labels* dialog box is displayed.



The contour label editor has the following options:

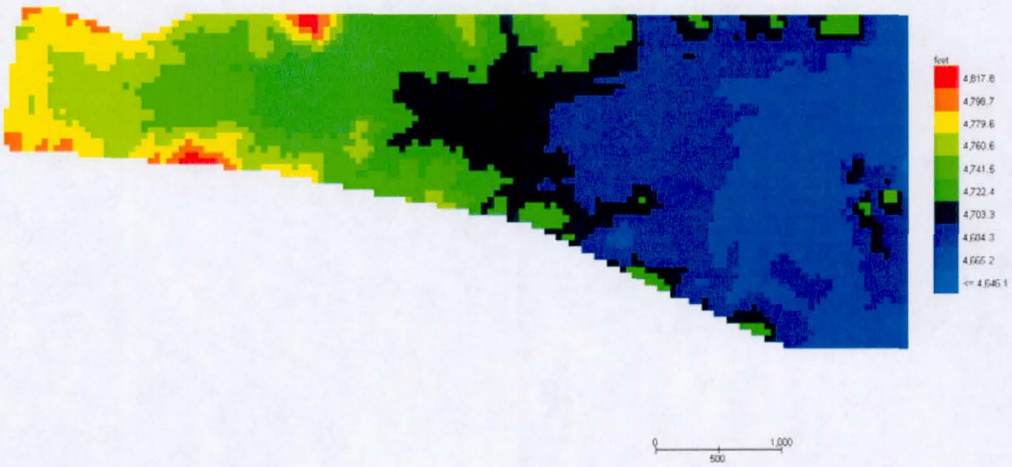
- *Horizontal alignment*: Text alignment in the horizontal direction and can be left, center or right justified.
- *Vertical alignment*: Text alignment in the horizontal direction and can be left, center or right justified.
- *Rotation*: Text rotation angle.
- *Labels with mask*: This option will create labels with a background *Mask color* to distinguish it from the contour lines.
- *Mask color*: Label background color.
- *Allow duplicates*: Allow more than one label for each contour line.
- *Font properties...button*: Displays Font customization dialog box to change font type, style, size, color, etc. as shown below:


Font
Font:
Times New Roman
Times New Roman
Trebuchet MS
Tunga
Tw Cen MT
Tw Cen MT Condense
Tw Cen MT Condense
Txt
Font style:
Regular
Regular
Italic
Bold
Bold Italic
Size:
10
10
11
12
14
16
18
20
OK
Cancel
Effects
Strikeout
Underline
Color:
Black
Sample
AaBbYyZz
Script:
Western

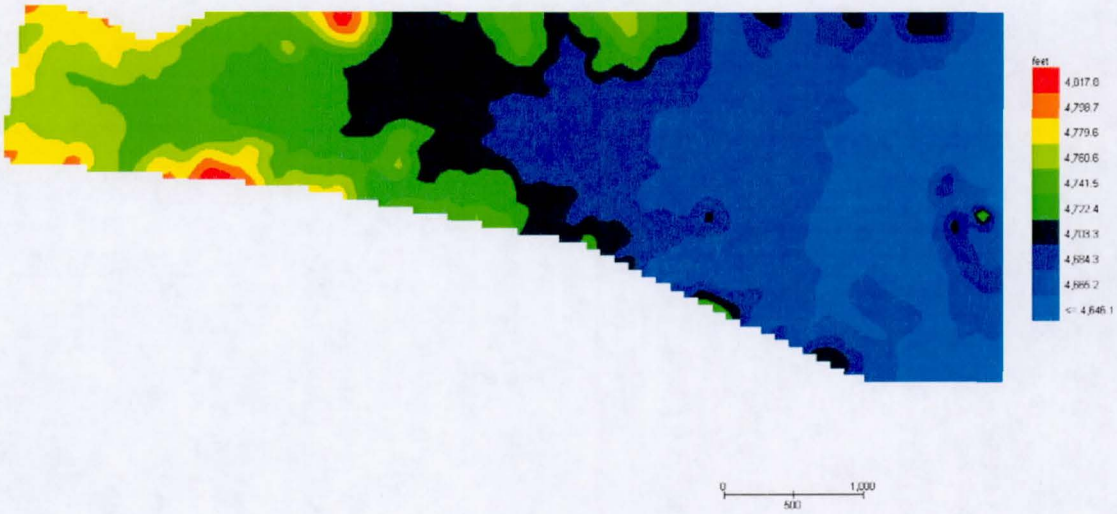
VII. Example Plots

The following are some examples of the *Mapper* plots.

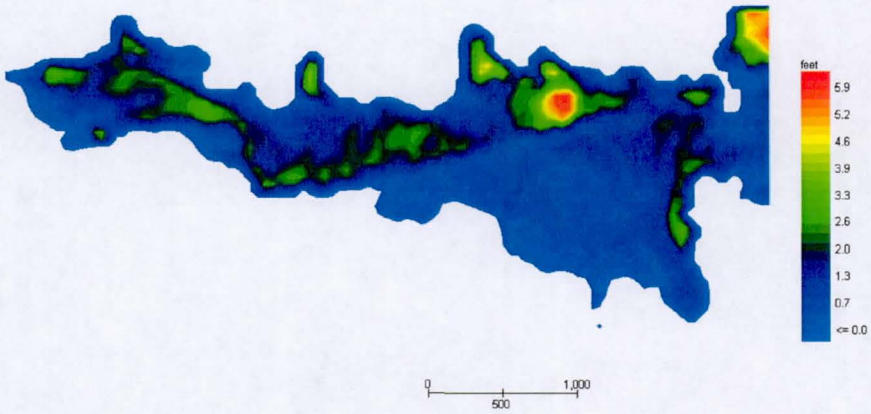
Grid Element Ground Surface Elevations



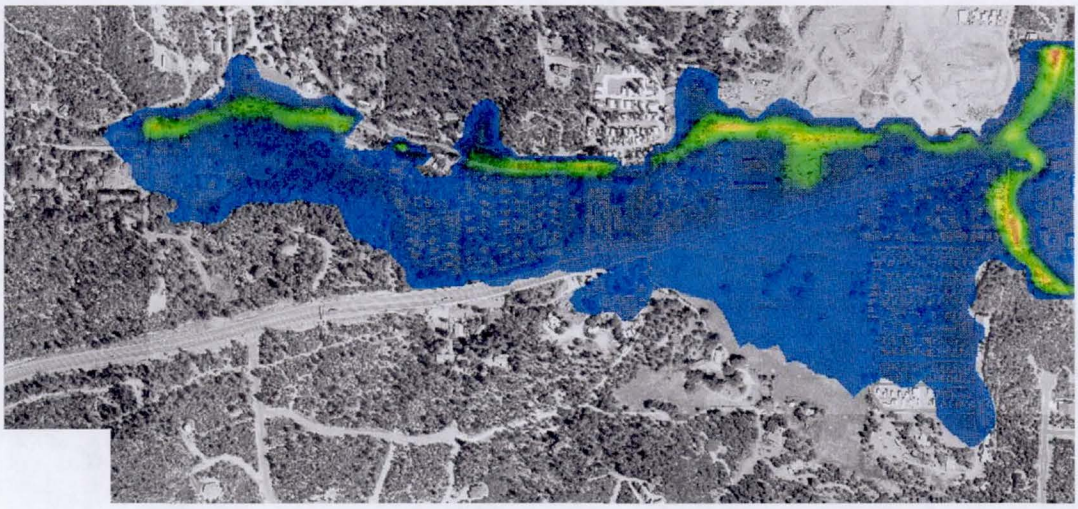
Shaded Contour Map of Grid Element Ground Surface Elevations



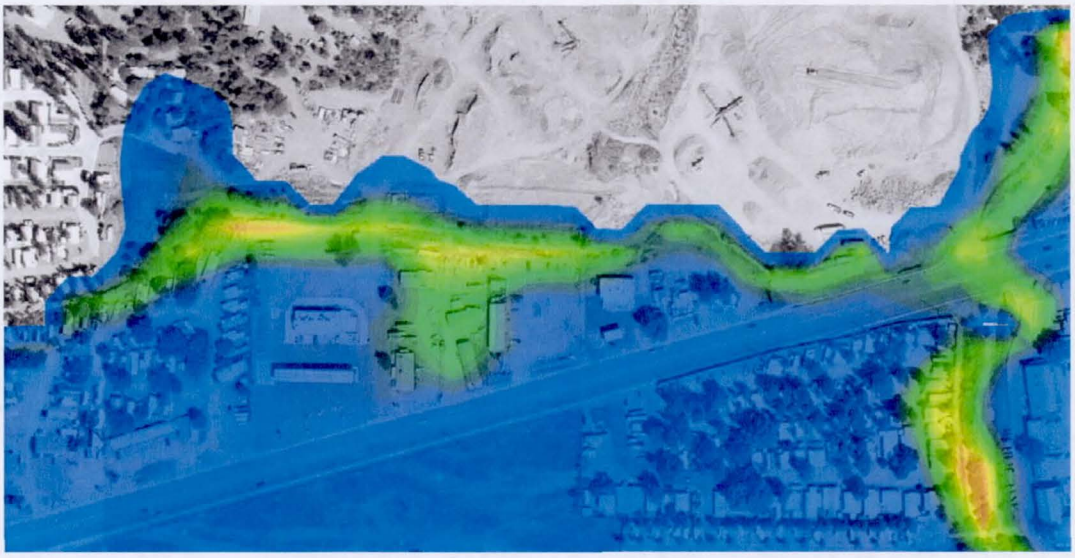
Shaded Contour Map of the Grid Element Maximum Flow Depth



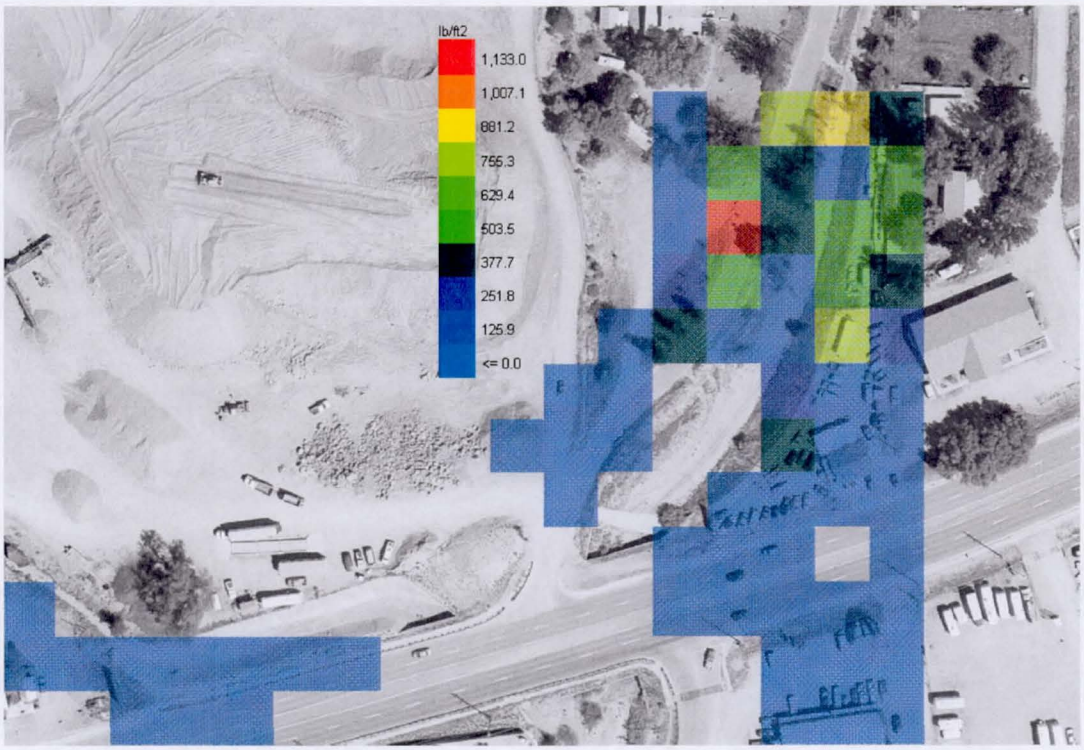
Shaded Contour Map of the Grid Element Maximum Flow Depth with Background Aerial Photograph



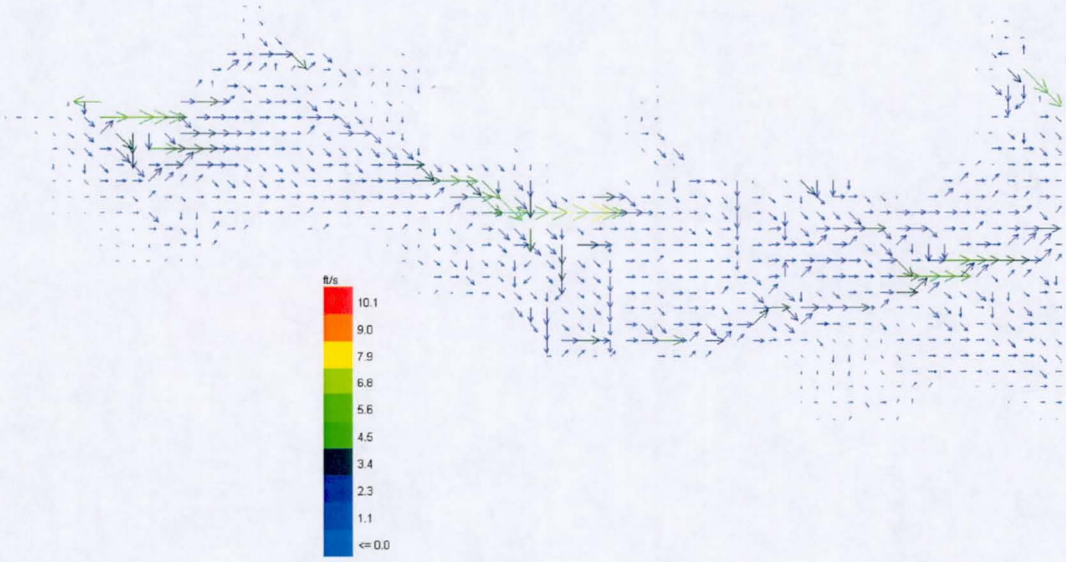
Zoomed Shaded Contour Map of the Grid Element Maximum Flow Depth with Background Aerial Photograph



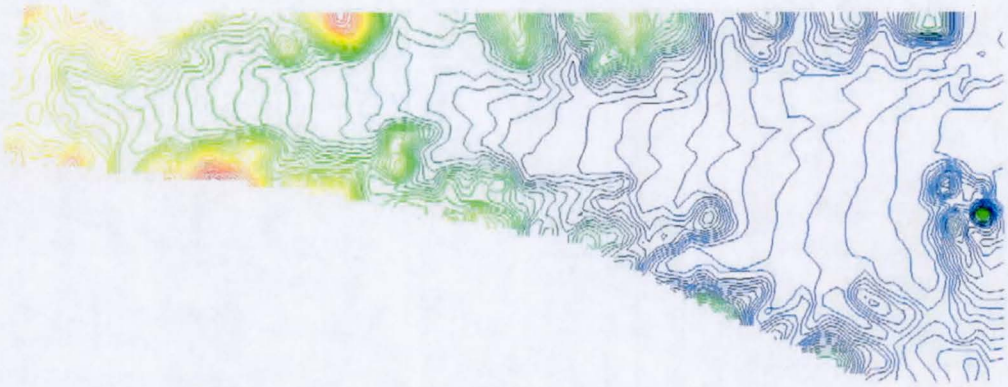
Grid Element Impact Force



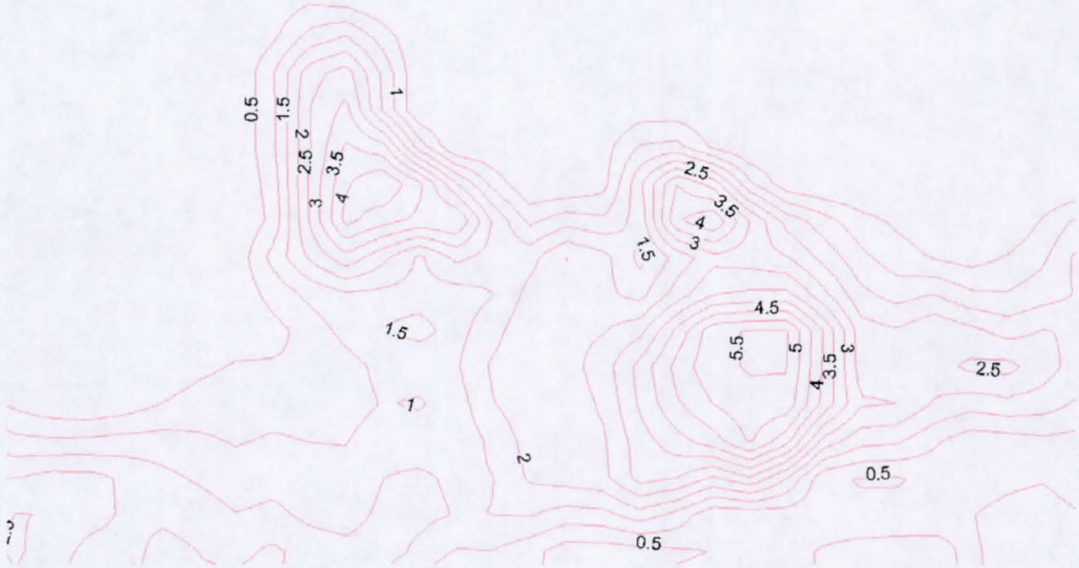
Grid Element Maximum Velocity Vectors



Line Contour Map of the Grid Element Ground Surface Elevations

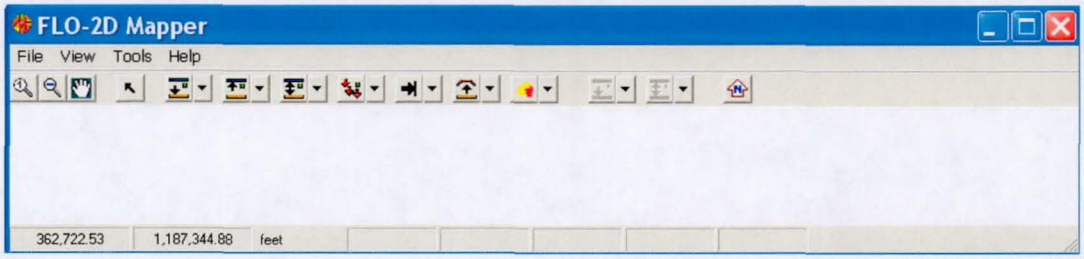


Line Contour Map of the Grid Element Ground Surface Elevations with Labels



VIII. Menus and Toolbar

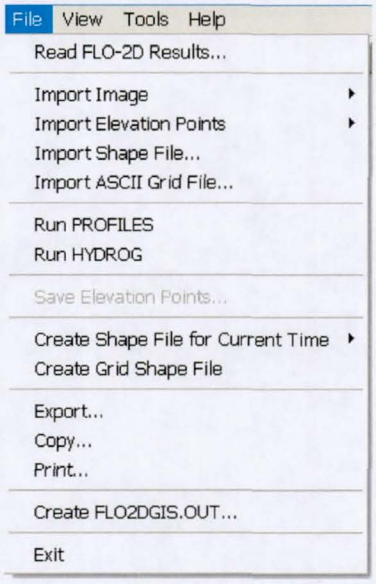
This section describes the *Mapper* commands in the main menu and the information toolbar:



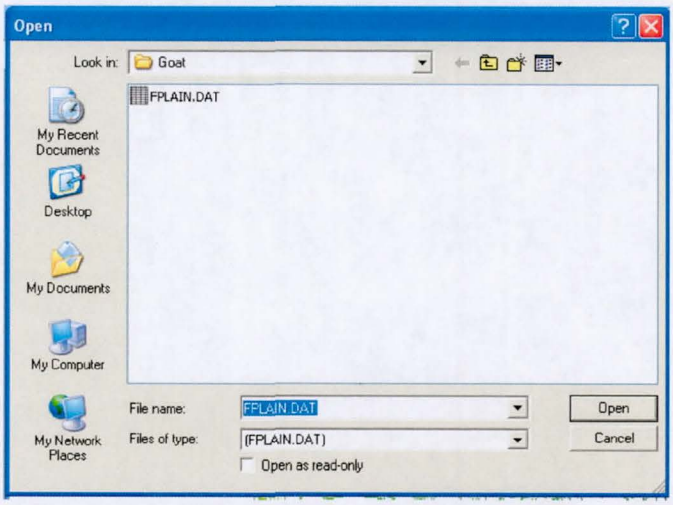
Each main menu command may contain one or more subcommands.

8.1 File Menu Commands

Read FLO-2D Results Command… (*File Menu*)

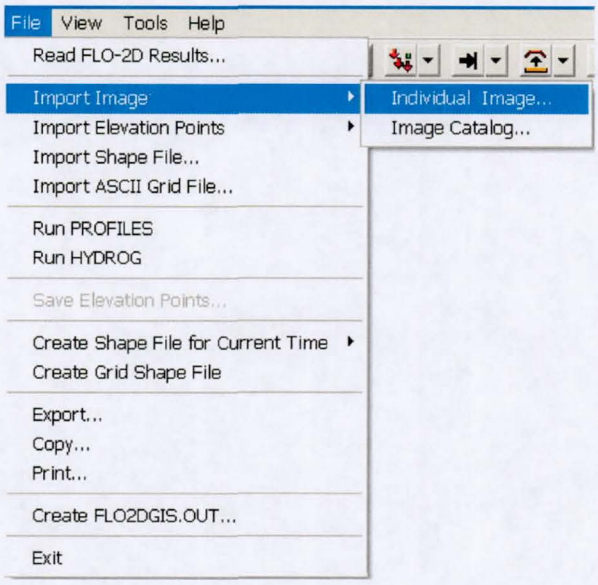


This command reads *FLO-2D* data files (*.DAT) and result files (*.OUT). A dialog box appears to allow selection of the FPLAIN.DAT file:



It is only necessary to select the FPLAIN.DAT data file. The other required and companion files are automatically read from the same directory.

Import Image/Individual Image (File Menu)



Use this command to import images such as aerial photos. Several images can be imported at the same time. To correctly position the image, each image must be accompanied by a "world" file containing its geo-referenced data. The world file must have the same file name as the corresponding image file and an extension name depending on the image type according to the table below. For example, if you have an image file named *myimage.bmp*, then the associated world file must be named *myimage.bmpw* or *myimage.bpw*.

Image File Extension	World File Extension
bmp	bmpw or bpw
jpg; jpeg	jpgw or jgw
tif; tff; tiff	Tfw
gis	Gsw
lan	Lnw
bil	Blw
bip	Bpw
bsq	Bqw
sid	Sdw
sun	Snw
rs; ras	Rsw
rlc	rcw

The world file has the following format:

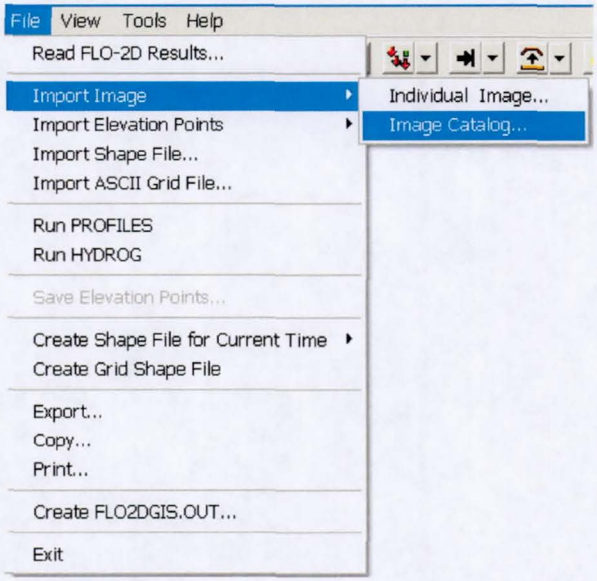
Line1: Dimension (width) of a pixel in map units in the x-direction (ft or m)
Line 2: 0
Line 3: 0
Line 4: Dimension (height) of a pixel in map units in the y-direction (ft or m). This value is negative because the image space is top-down, whereas map space is bottom-up.
Line 5: x-origin (x-coordinate of the center of the upper left pixel)
Line 6: y-origin (y-coordinate of the center of the upper left pixel)

Example:

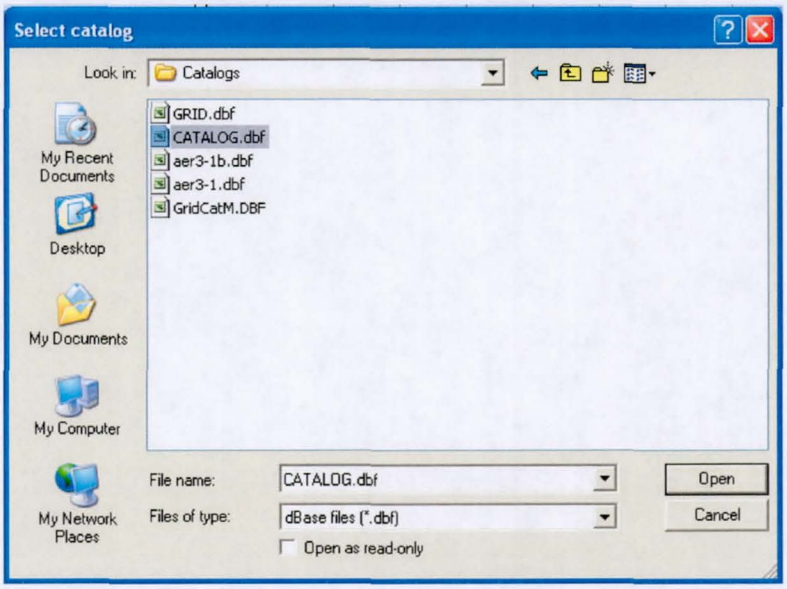
```
20
 0
 0
-20
637510
1032490
```

In this example the pixel width and height is 20 ft and the coordinate of the center of the upper left pixel is 637510, 1032490. To determine the pixel size of an image you can use an image editing program. For example in MS-Paint load the image and use the Image/Attribute command to query the number of pixels in x and y directions. Then divide the x and y widths of your image in real coordinates by the number of pixels in each direction.

Import Image/Image Catalog (File Menu)



This command allows importing of several image files stored in any subdirectory. First draw a polygon on the grid and then select an image catalog file.



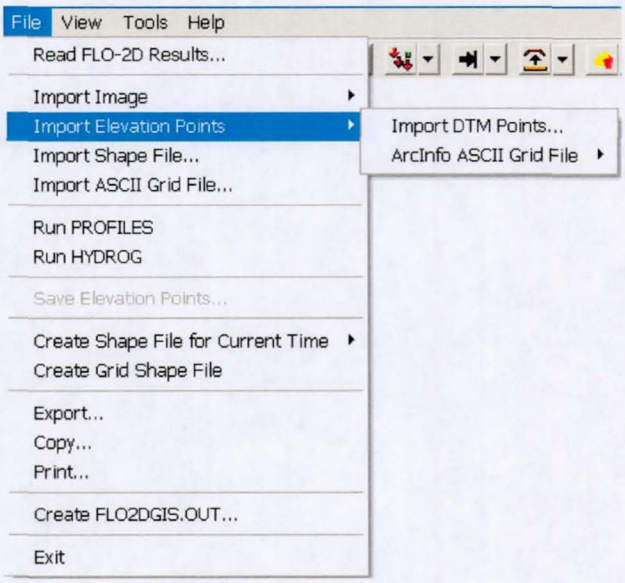
The catalog file may be in DBASE or ASCII format and has the following format:

IMAGE	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\ 6401030-5.TIF	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\ 6401035-5.TIF	660000	1300000	770000	1500000
\\Agua\IMF\Publico IMF\ 6401055-5.TIF	630000	1300000	750000	1510000

The first column is the file name including its path and the following four columns are the image limit coordinates. *Mapper* will find all images from the catalog that are contained or intersect the

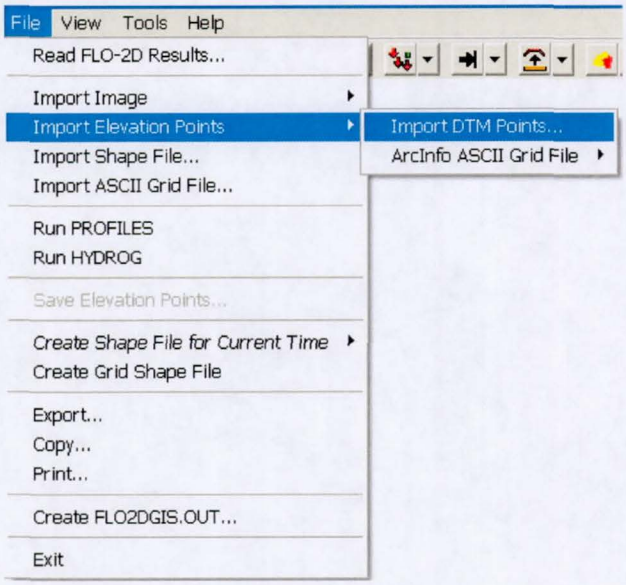
user defined polygon and will retrieve the appropriate ones.

Import Elevation Points Command (File Menu)



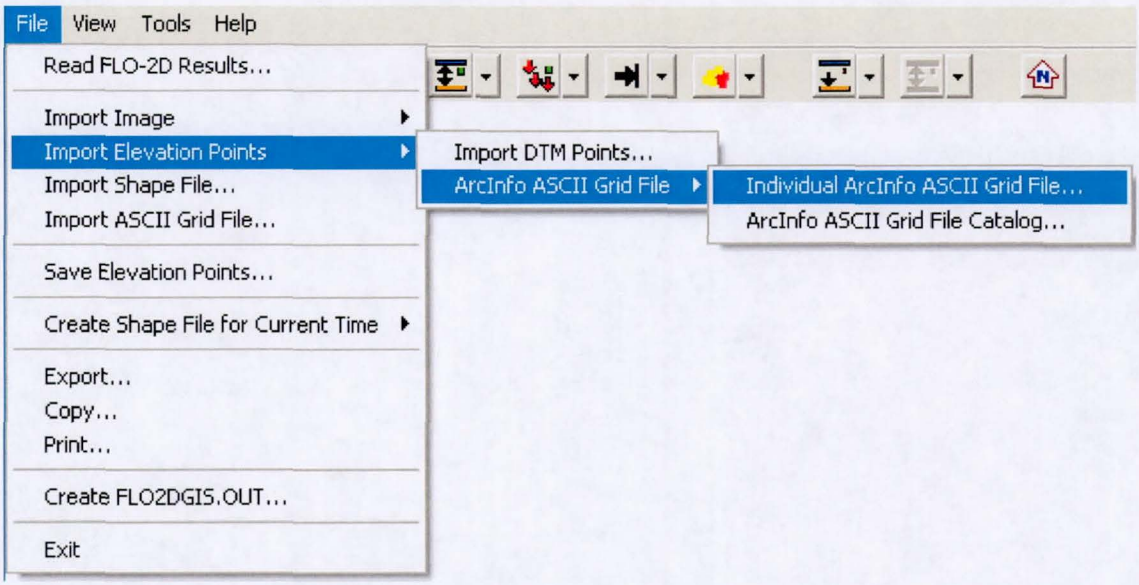
This command will import an existing digital terrain elevation file. There are two submenu commands to import either digital terrain elevation points (DTM Points Command) or ArcInfo ASCII Grid File.

Import DTM Points...Command (File Menu)



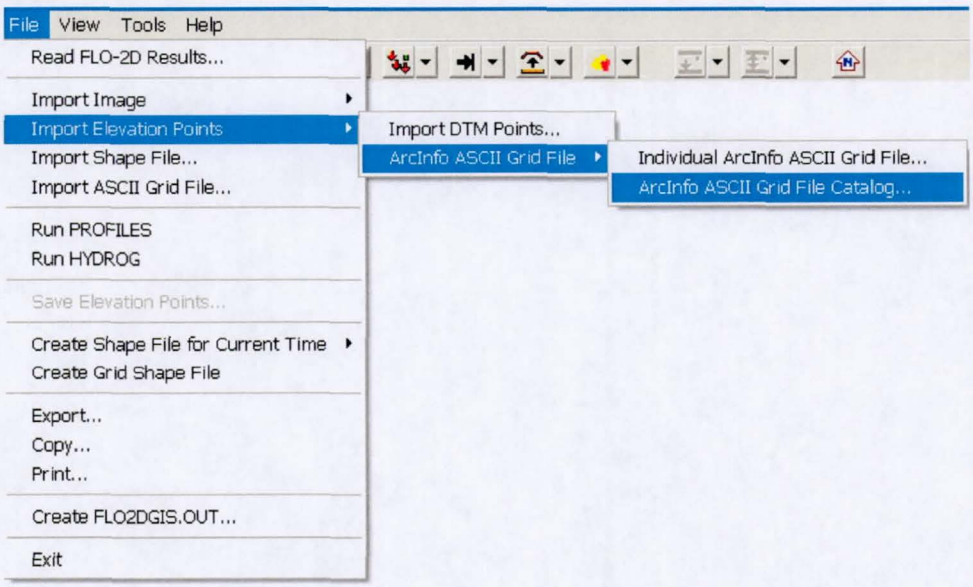
This command imports DTM elevation points from an existing file. You may import several data files. The new points are added to the existing data. You may also mix or combine DTM points with other random elevation points from an ArcInfo ASCII grid files.

Individual ArcInfo ASCII Grid File Command (File Menu)

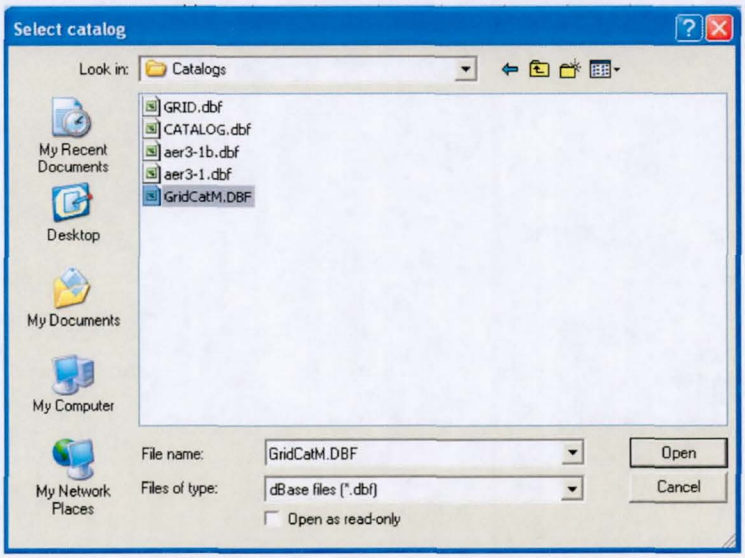


Use this command to import an individual ArcInfo ASCII grid files. Several grid files can be imported. The new points are added to the existing data. This data can also be mixed or combined with the DTM points.

ArcInfo ASCII Grid File Catalogue (File Menu)



This command will import several ArcInfo ASCII grid files stored in any subdirectory. First draw a polygon on the working region or grid system and then select a catalog file.

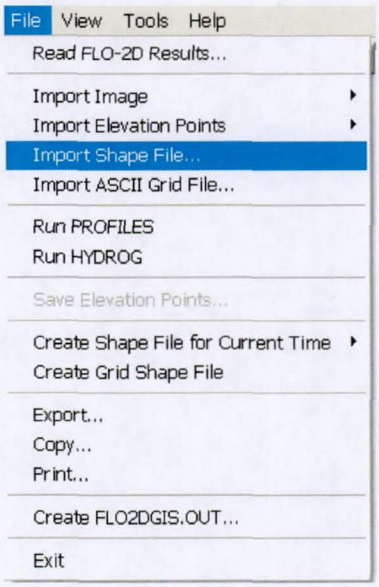


The catalog file may be in DBASE or ASCII format and has the following format:

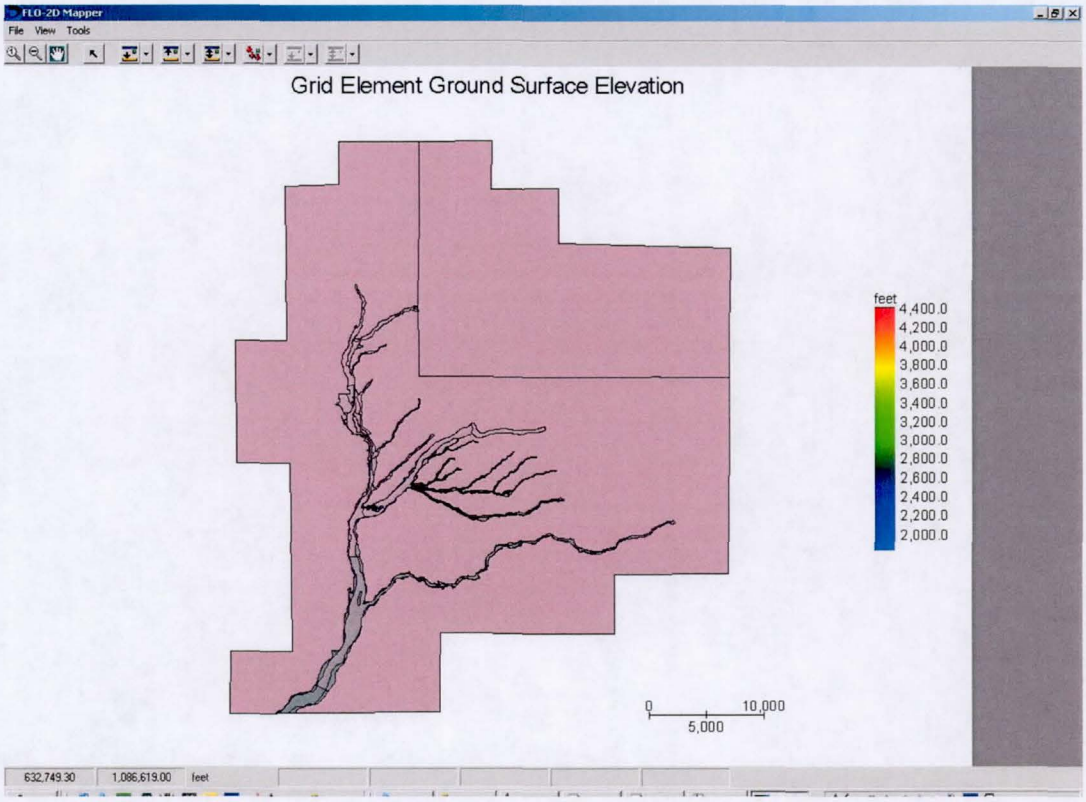
ASCII_GF	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\grd2375-100.asc	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\grd2376-100.asc	500000	1000000	900000	1500000
\\Agua\IMF\Publico IMF\grd2476-100.asc	510000	1000000	740000	1510000

The first column is the file name including its path. The following four columns are the data limits in each file. *Mapper* will find all data files that are contained or intersect the user defined polygon and retrieve the relevant files.

Import Shape File Command (File Menu)



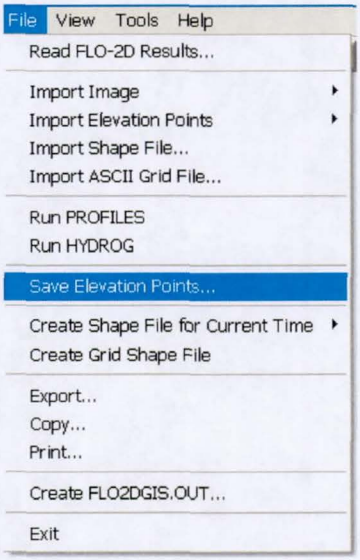
This command will import ESRI shape files. The shape file data map can be related to soil properties, land use, vegetation, Manning roughness coefficients or similar data. Once read, the shape file is displayed in the screen like the following example from Maricopa County Manning coefficients shape file.



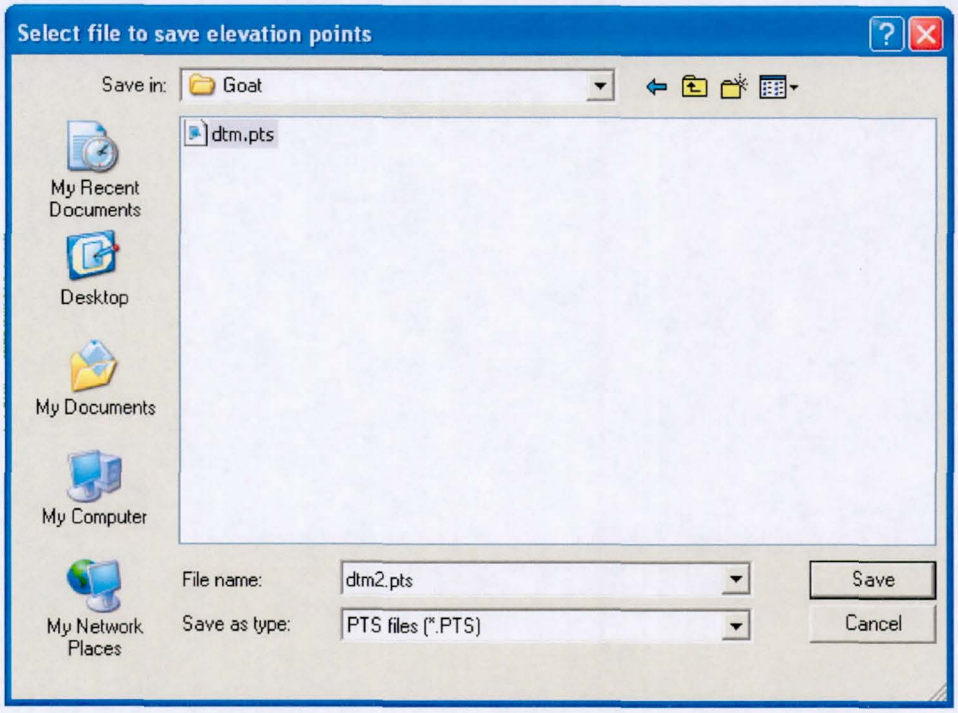
Import ASCII Grid File Command (File Menu)

This command will import ArcInfo ASCII grid files stored in any folder.

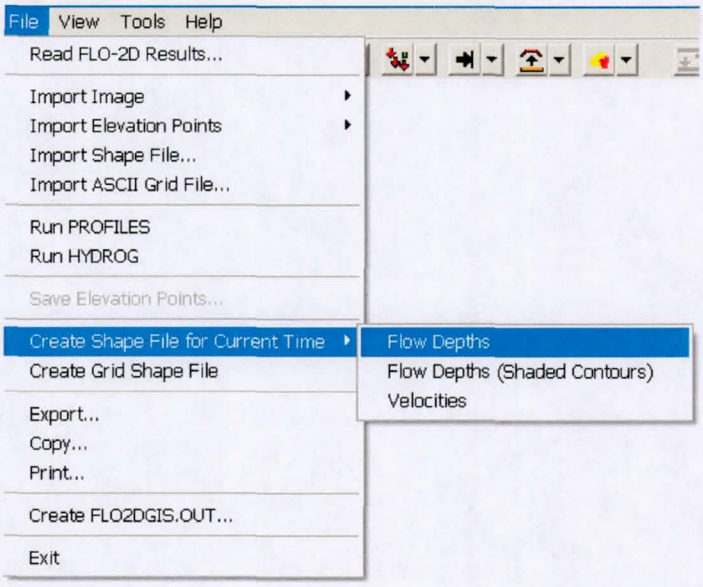
Save Elevation Point Command (File Menu)



This command is used for saving the DTM elevation points to an ASCII file. You may have imported a set of DTM points and then deleted selected points that are outside the computational grid system. With this command you can export save the remaining DTM points in a file. After clicking the *Save Elevation Points…* command, the following dialog enables you to define the DTM point ASCII file:



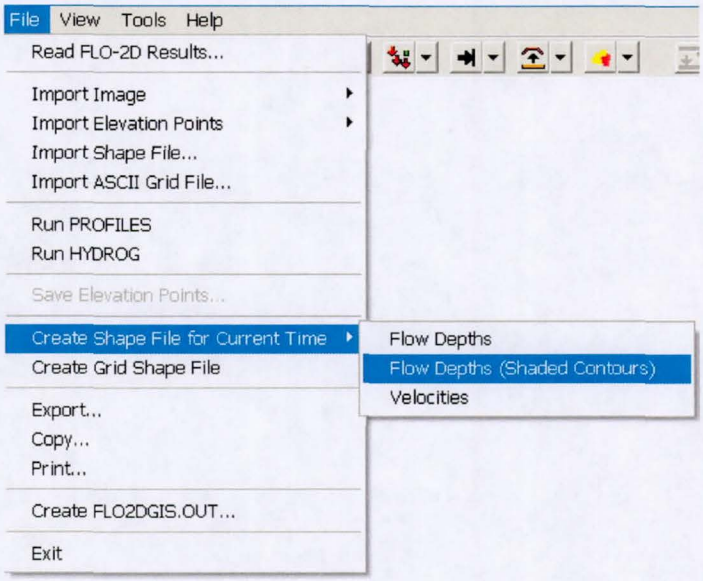
Create Shape File for Current Tim/Flow Depths (File Menu)



This command will export an ESRI polygon Shape File with the current grid element based flow depth time plot shown in the animation. The shape file name will indicate the time and

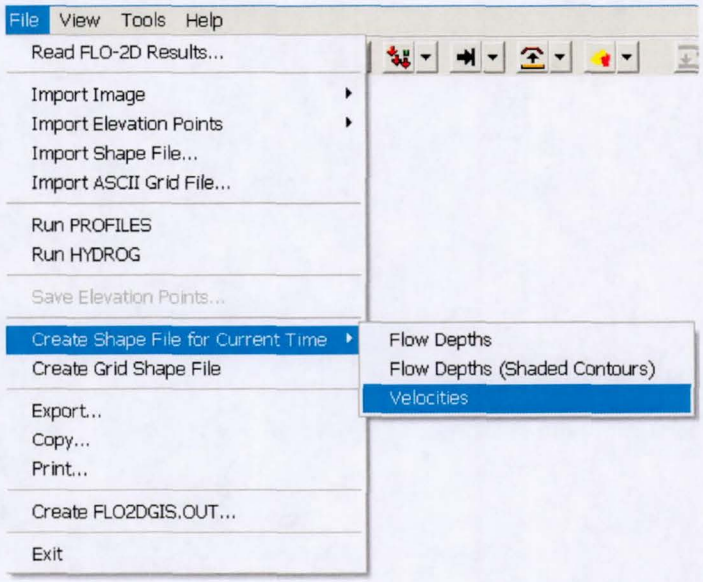
that it is a grid element flow depth plot.

Create Shape File for Current Tim/Flow Depth (Shaded Contours) (File Menu)



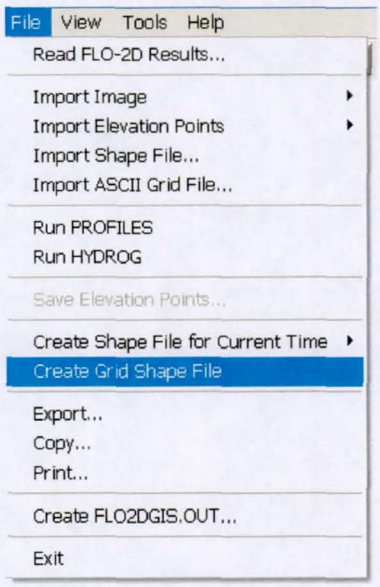
This command will export an ESRI polygon Shape File with the current flow depth time shaded contour plot constructed from the results shown in the animation. The shape file name will indicate the time and that it is a flow depth shaded contour plot.

Create Shape File for Current Tim/Velocities (File Menu)



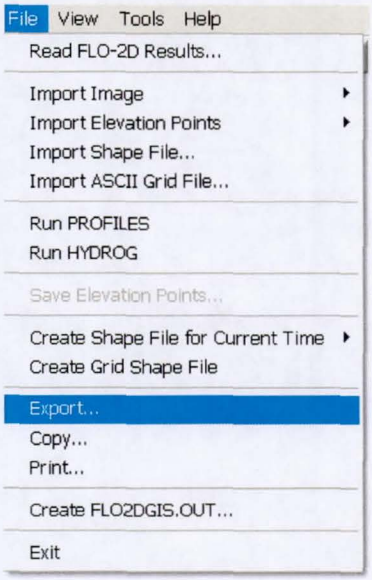
This command will export an ESRI polygon Shape File with the grid element velocities for the current time plot shown in the animation. The shape file name will indicate the time and that it is a grid element velocity plot.

Create Grid Shape File (File Menu)

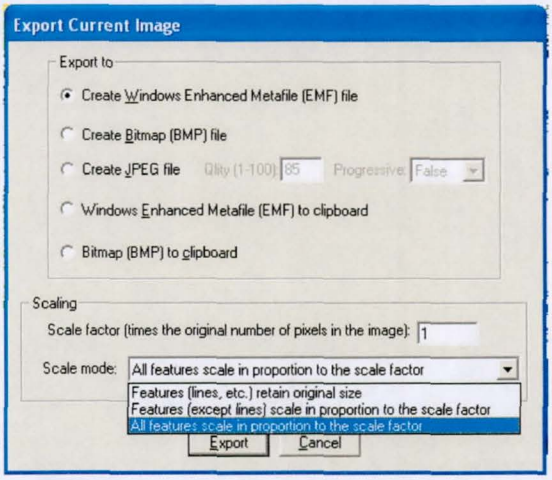


This command will create a polygon shape file named *MGrid* that has the grid element numbers as attributes for each grid element.

Export...Command (File Menu)



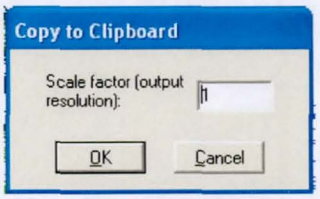
Using this command you can export the current screen view to a different image format. Click on the Export Command the following dialogue box will appear:



With this dialog box, you can select various image export formats such as Bitmap, JPEG, Windows Metafile, etc. The scale factor of the image can be adjusted to improve print quality of the image. When you click on the Export button, input the image file name and select the directory to save the image in the subdirectory/file dialogue box.

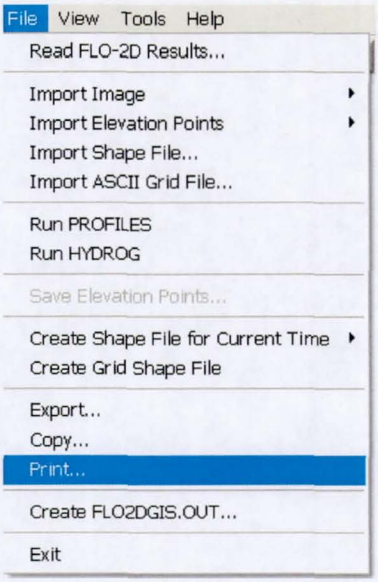
Copy... Command (File Menu)

Use this command to copy the current plot to the clipboard. The plot may then be pasted into other documents. When you select the copy command the following dialog appears:



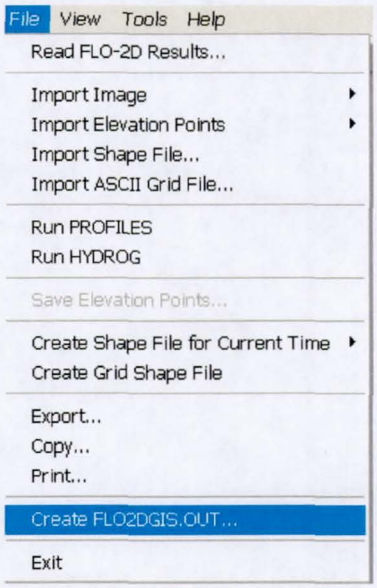
The default *Scale factor* is 1, but you may set higher values to improve output resolution. It is not recommended to use values higher than 3, since plots with this resolution require substantial amount of memory that may result in a very slow print process.

Print... Command (File Menu)



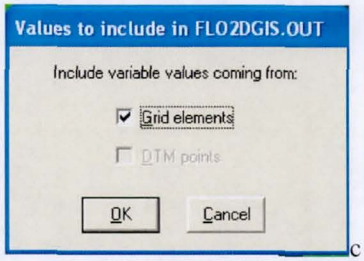
Use this command to print the current plot. When you select this command, the standard printer management dialog allows you to choose the printer.

Create FLO2DGIS.OUT... Command (File Menu)



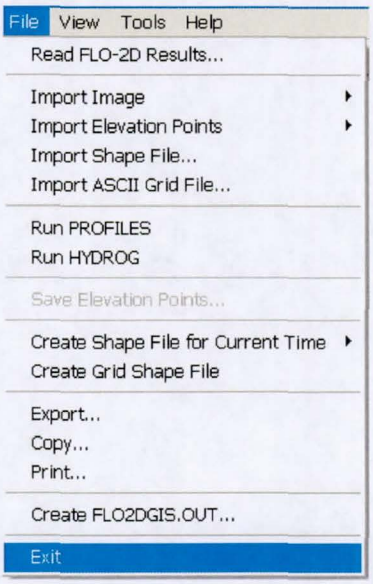
Use this command to create a GIS output file. The FLO2DGIS.OUT file contains the x- and y-coordinates of each DTM point and their associated ground elevation, water surface elevation, flow depth and water velocity. It is necessary to first interpolate the flow depth by

subtracting the DTM ground points from the grid element water surface elevation. This is accomplished by using the *Interpolate to DTM Points* command in the Tools Menu. This GIS output file can be imported to ArcView™. *Mapper* displays a dialog box to select the values to be written into the file:



The flow depth values assigned to the DTM points can only be written to the FLO2DGIS.OUT file if the flow depth interpolation by subtracting the DTM point ground elevations from the grid element water surface has been completed.

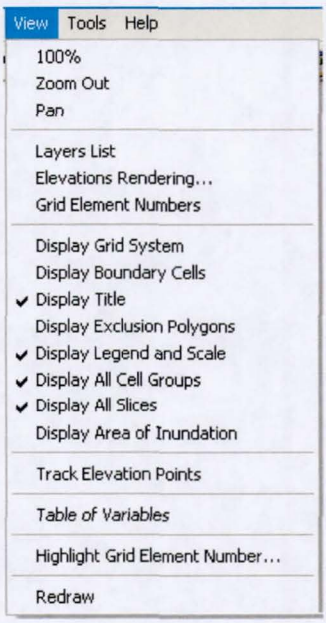
Exit (File Menu)



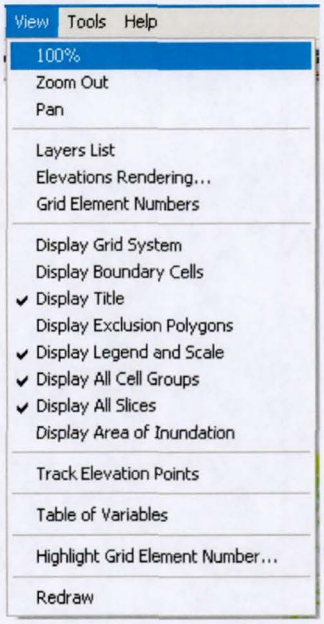
This command ends the *MAPPER* session. You can also use the *Close* command on the application Control menu.

Shortcuts: Mouse: double-click the application Control-menu button.
Keys: Alt-F4

8.2 View Menu Commands

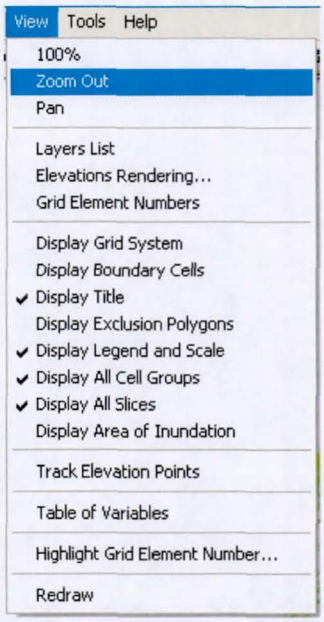


100% Command (View Menu)



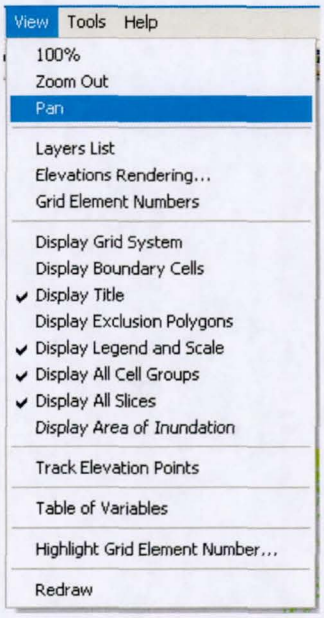
Use the *100%* command to reset the working region to its original size. The *100%* toolbar icon can also be used. To zoom-in (magnification of the working region) just left-click in the working region and drag the mouse for magnification desired.

Zoom Out Command (View Menu)



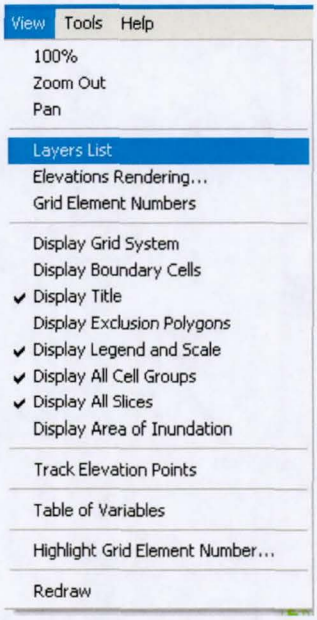
Use the *Zoom Out* command to return the working region to its previous zoom extent.

Pan Command (View Menu)



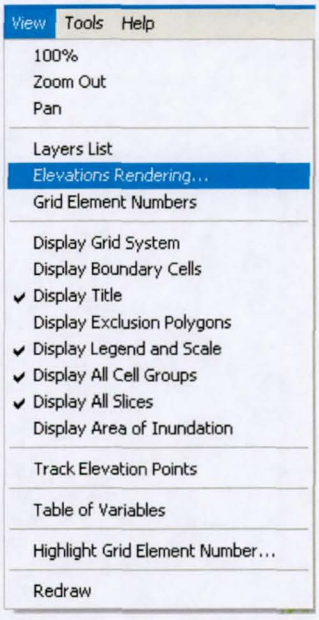
Use the *Pan* command to move around and redisplay the working region view. When activated the mouse cursor changes to *Pan* mode and you can drag and click to pan around. The *Pan* toolbar icon achieves the same result.

Layers List Command (View Menu)



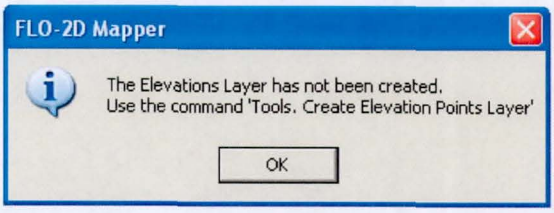
This command is used to manage existing layers. Please refer to GDS Layer List Command for instructions. The commands are identical.

Elevation Rendering... Command (View Menu)

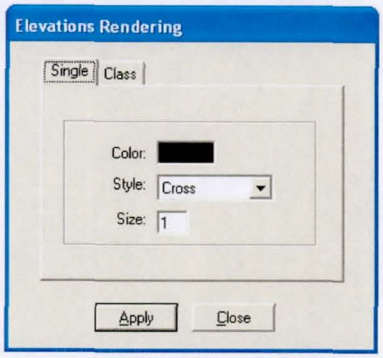


This command may be used to select how the elevation points are displayed.

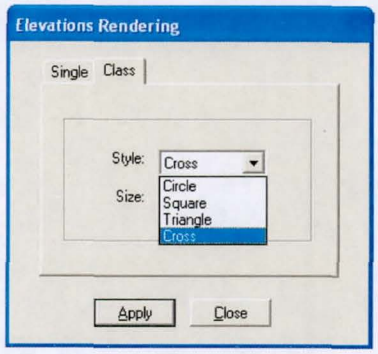
To optimize display times, *Mapper* does not automatically create the Elevation layer. For certain functions like the *Elevation Rendering* this layer is required. If the layer does not exist, you will get the following message:



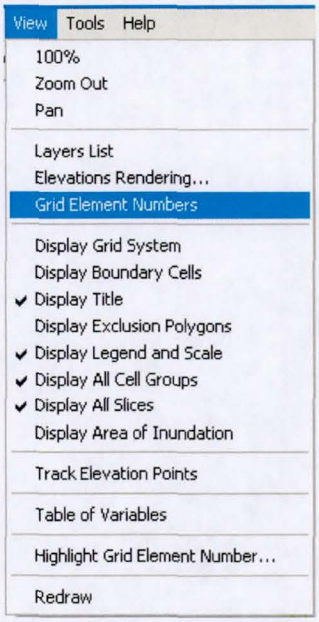
To create this layer use the *Create Elevation Layer* command in the Tools menu. After the elevation layer is created, click on Elevation Rendering again to get the dialog:



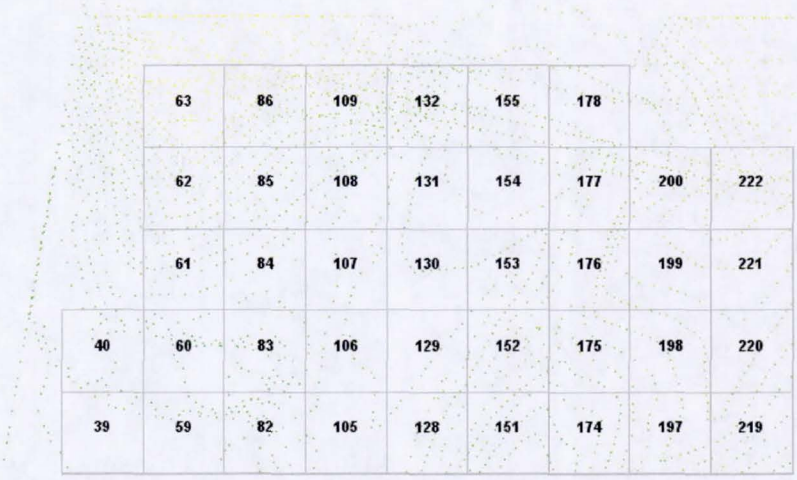
The "Single" tab selects a single color, style, and size for all elevations. The "Class" tab selects a style (point descriptor) and size for the points. A palette of 256 colors are available to be assigned to the point elevation.



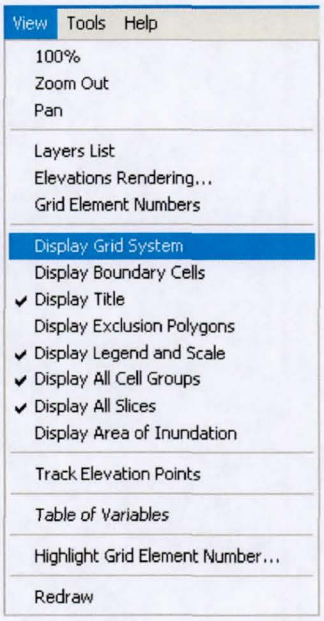
Grid Element Numbers Command (View Menu)



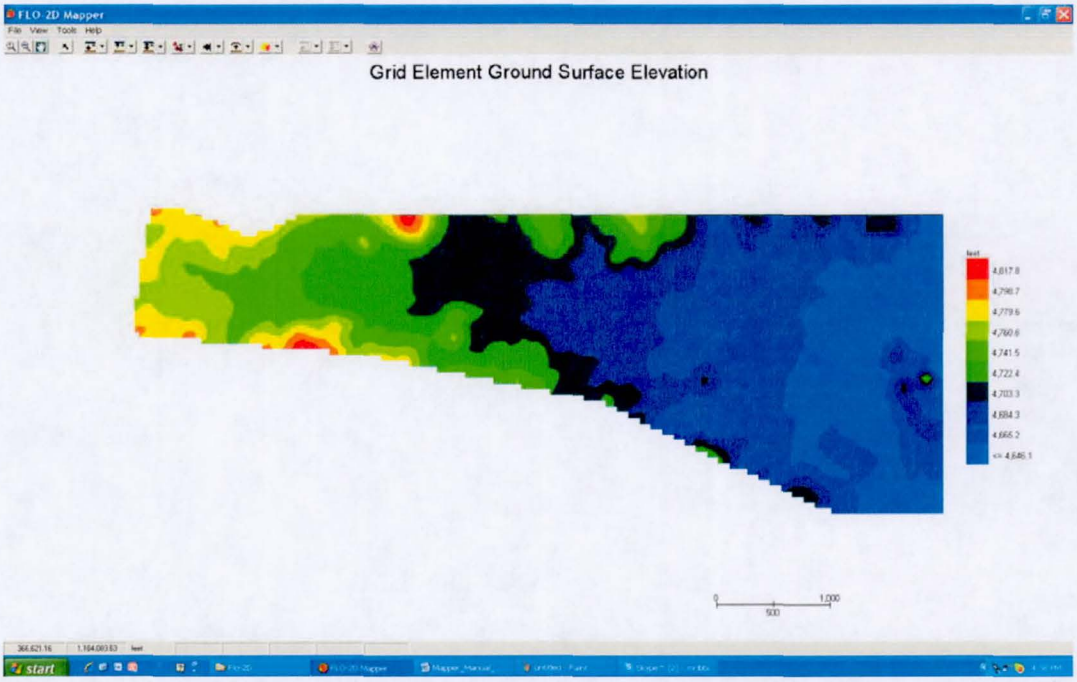
By default *Mapper* does not display grid element numbers. Use this command to display grid element numbers. Note that if the grid system is very large, the higher number grid element numbers may not fit into the elements. Zoom in to enlarge the view and allow clear view of grid element numbers.



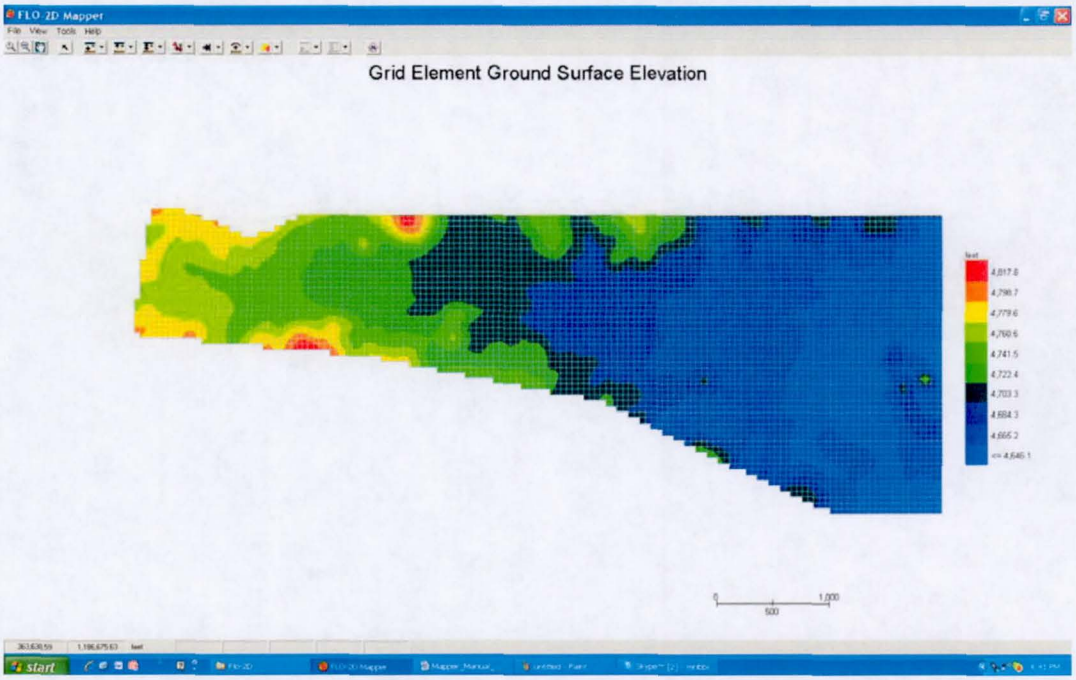
Display Grid System Command (View Menu)



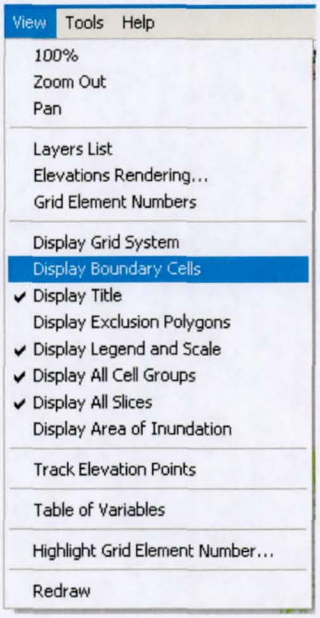
Click on this command to view the *FLO-2D* grid system as an overlaid image. The following plot does not display the grid system:



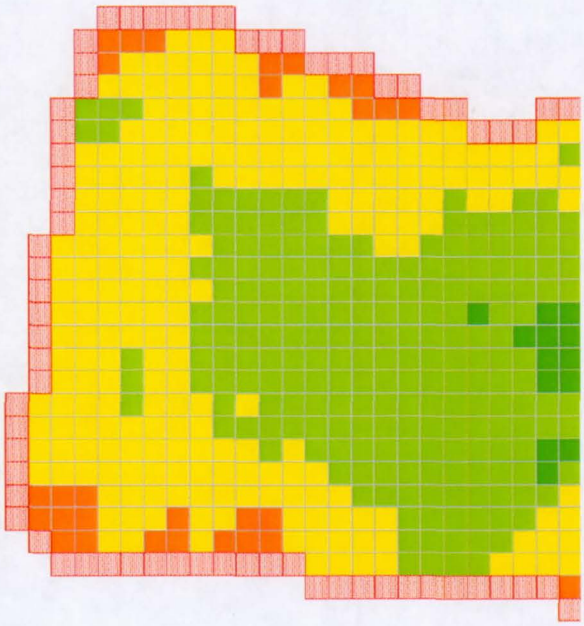
The following plot does displaying the grid system:



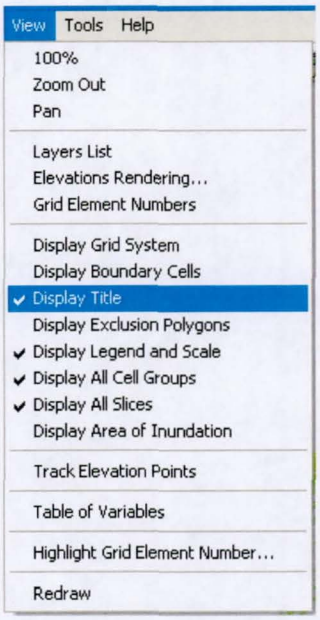
Display Boundary Cells Command (View Menu)



This command displays the *FLO-2D* grid boundary cells.

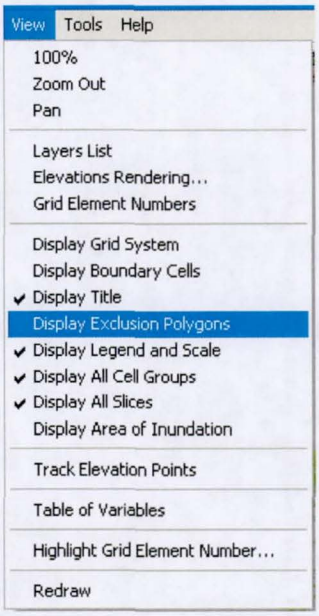


Display Title Command (View Menu)



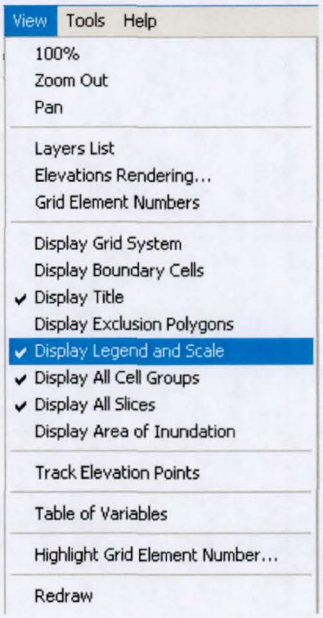
This command allows you to display or hide the plot title.

Display Exclusion Polygons Command (View Menu)



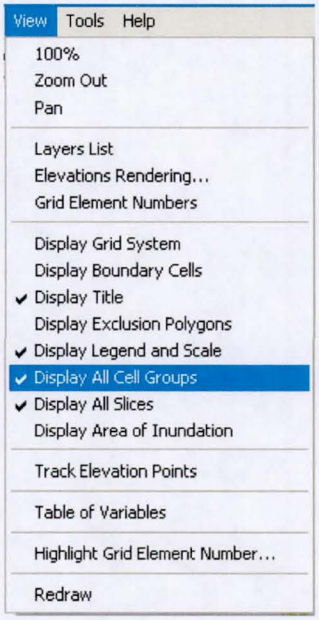
This command allows you to display or hide the polygons used to delete unwanted parts of shaded contour plots based on DTM data. To see Exclusion Polygons they have to be created using the *Tools/Exclusion Polygons/Create Exclusion Polygons*.

Display Legend and Scale (View Menu)



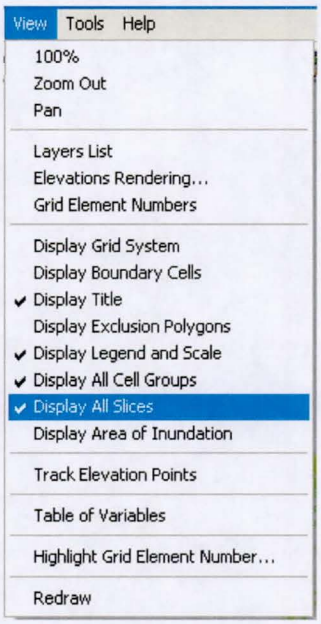
For the various flood variable plots, this command displays the map legend and scale.

Display All Cells Groups (View Menu)



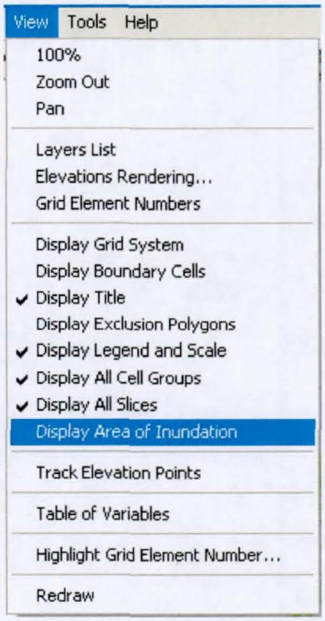
This command highlights all grid elements (cells) that are being used for plotting Flow depth and velocity vs. time plots.

Display All Slices (View Menu)



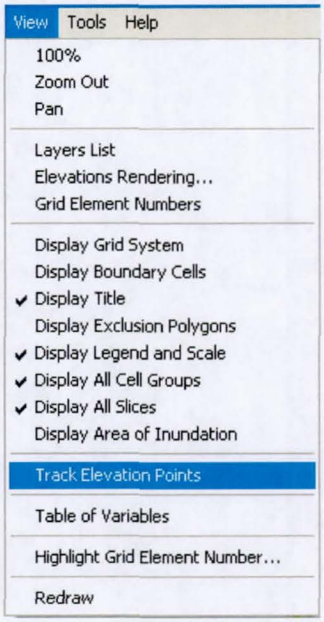
This command highlights all slices used for plotting flow depth and bottom eleveation profiles.

Display Area of Inundation (View Menu)



This command highlights the contour that limit the area inundated simulated by the FLO-2D model. This plot must have been previously created using the *Create Area of Inundation Layer* in the *Tools* menu.

Track Elevation Points Command (View Menu)



With this command, the elevation of any DTM point is displayed in a dialog box. When the mouse pointer is placed in the vicinity of a point, its elevation is displayed in the right text box of the bottom toolbar:

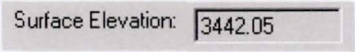
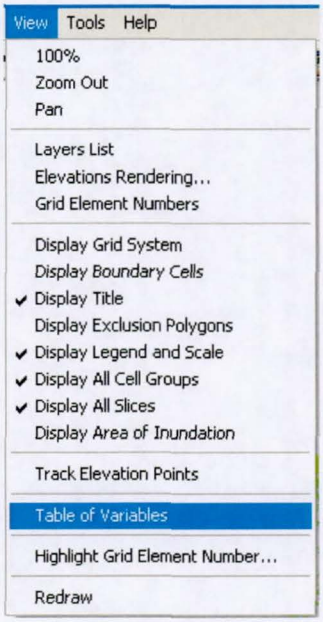
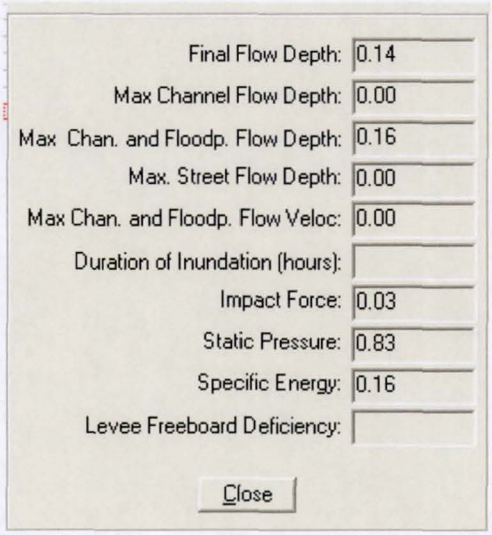


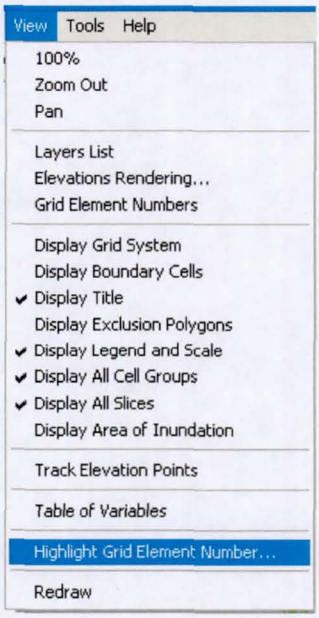
Table of Variables (View Menu)



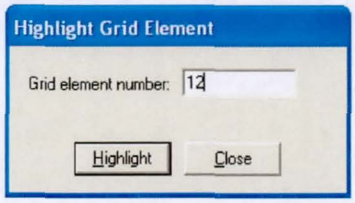
This command displays a floating window with a table of FLO-2D hydraulic output data. When you move the mouse over the grid, the values are displayed for each grid under the cursor.



Highlight Grid Element Number... (View Menu)

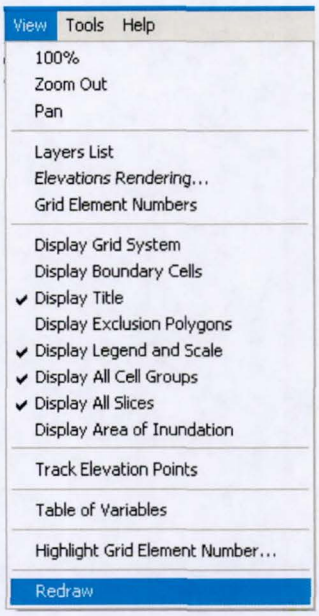


This command enables you to enter a grid element number in the following dialog box and locate it in the FLO-2D grid system. *Mapper* will prompt for the grid element number:



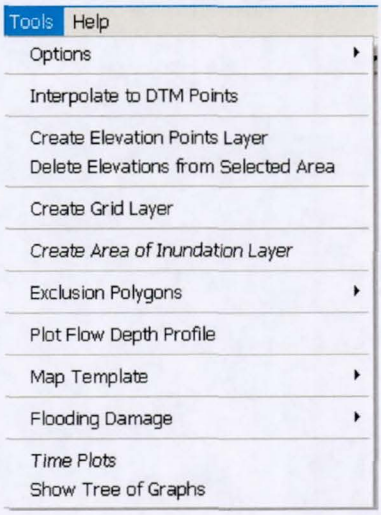
When you click the *Highlight* button, the grid element will blink. Note that if the selected element number is not in the current view, you may have to zoom out to view the entire grid system in order to locate the grid element.

Redraw Command (View Menu)

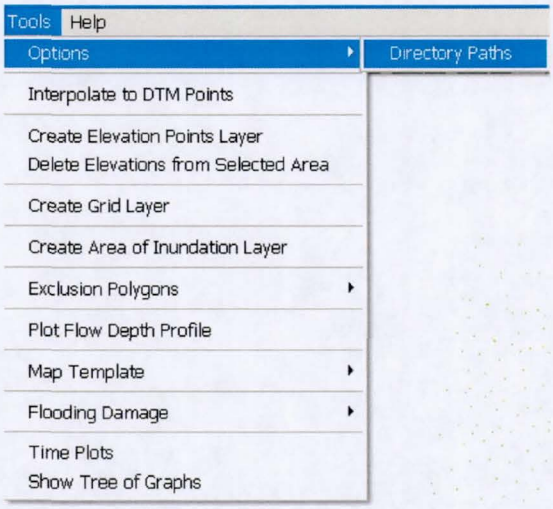


Use this command to redraw the visual objects in the *working region*.

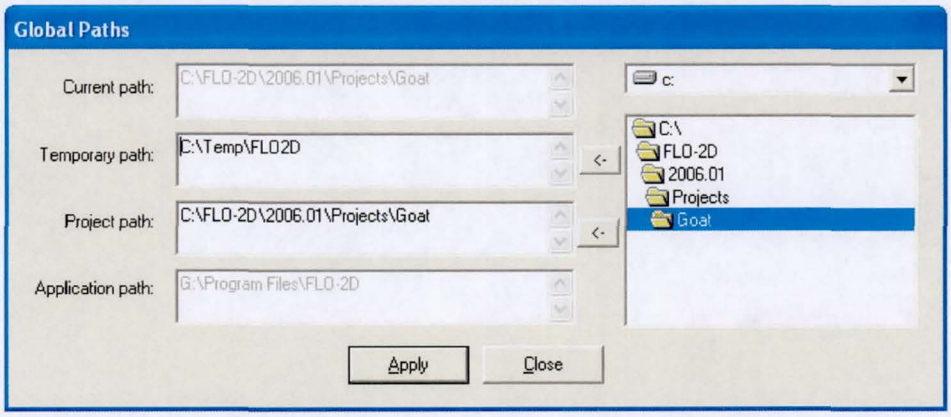
8.3 Tools Menu Commands



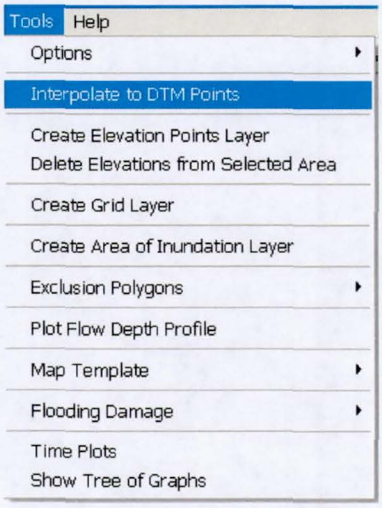
Option Directory Paths Command (Tools Menu)



This command allows you to change the *Mapper* default path by editing the settings in the following dialog box:

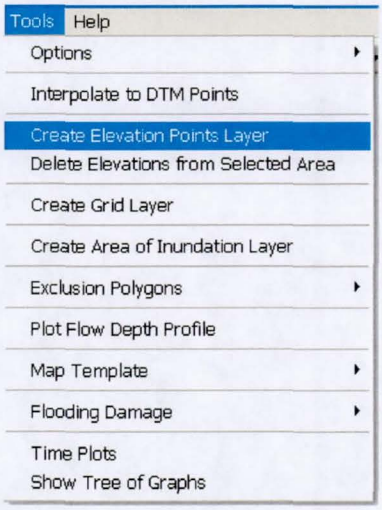


Interpolate to DTM Points Command (Tools Menu)



The purpose of this command it to create a TIN (Triangulated Irregular Network) based on the flow depths at each DTM point. The TIN is later used to construct the DTM Max Flow Depth Shaded Contour. To use this tool you first need to import the DTM points using the *Import DTM Elevation Points* command in the *File* menu. When this command is activated, the flow depth at each DTM point is computed by subtracting the DTM ground surface elevation from the grid element water surface elevation. You can import the DTM point file that was originally used in the *GDS* to interpolate grid element elevations.

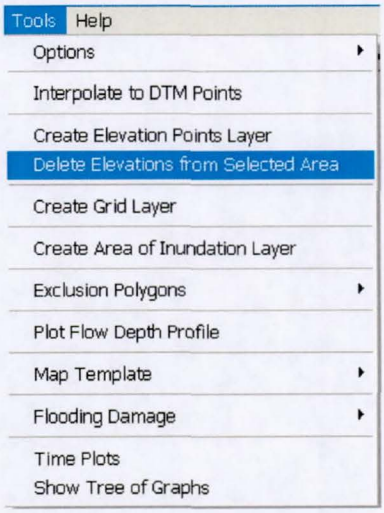
Create Elevation Points Layer Command (Tools Menu)



To optimize display times *Mapper* does not automatically create the Layer that contains the DTM Elevation points. For certain functions in *Mapper* this layer is required. If you try to perform to initialize a component in *Mapper* that requires this layer and it does not exist, the following message will appear:

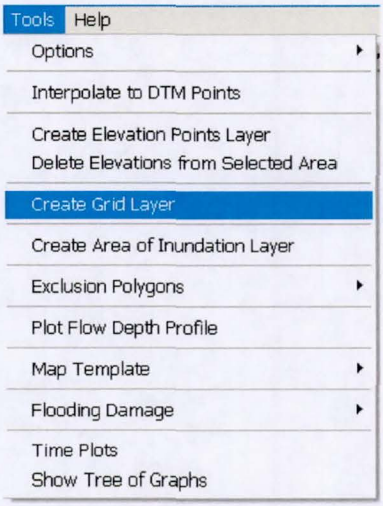
This command will create the Elevation Point Layer.

Delete Elevations From Selected Area Command (Tools Menu)



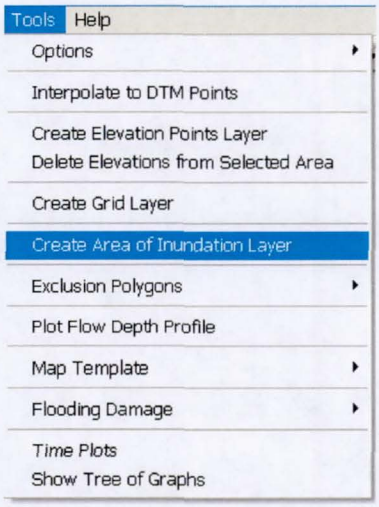
This command will delete unnecessary DTM elevation points from the working region. The points are deleted by enclosing them in a polygon. The dialog box will appear instructing you to draw a polygon around the area will the DTM points that will be deleted. Close the polygon with a double click and another dialog box will be displayed asking you to confirm the points deletion. You can save the edited points file using the *Save Elevation Points*...command in the *File* menu.

Create Grid Layer Command (Tools Menu)



This command creates the Grid Layer that is necessary for some operations performed on the grid.

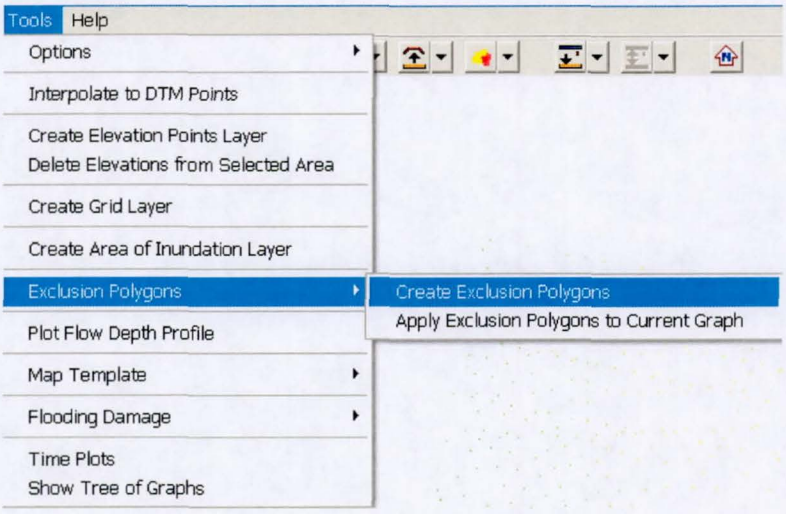
Create Area of Inundation Layer Command (Tools Menu)



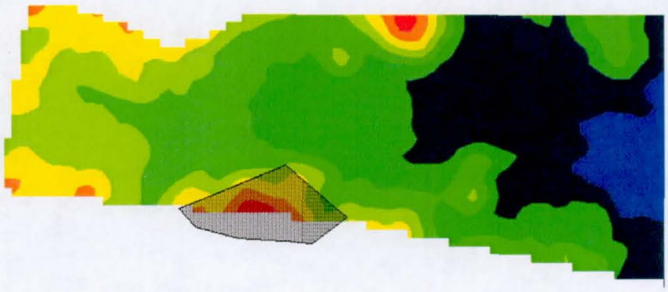
This command creates contour lines that enclose the areas inundated as simulated by the FLO-2D model.

Exclusion Polygons Command (Tools Menu)

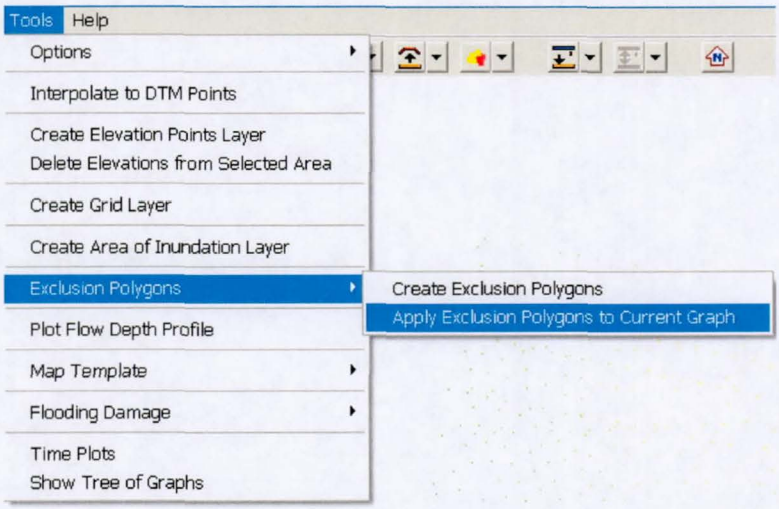
This function will erase unwanted areas from line or shaded contour plots. This is a useful tool when *Mapper* creates contour plots in convex regions or other areas where the contour lines have spilled over physical features should as levees or hillsides. This may occur because of either a grid element may straddle a physical feature or because there are DTM points located on the other side of the feature. It may also occur because of the finite thickness of contour lines and shade contour outlines. This tool allows you to delete the contour graphics inside an exclusion polygon. First click on the *Create Exclusion Polygon*:



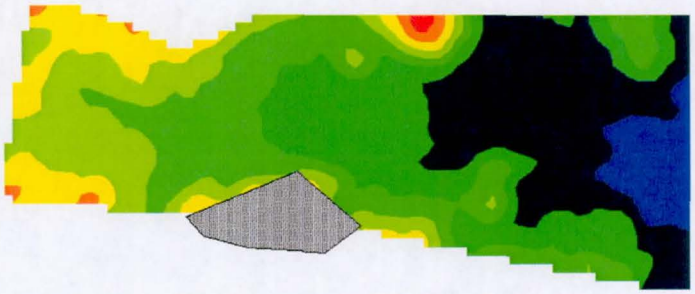
Then draw the polygon area that you want to erase. *Mapper* will hatch the polygon:



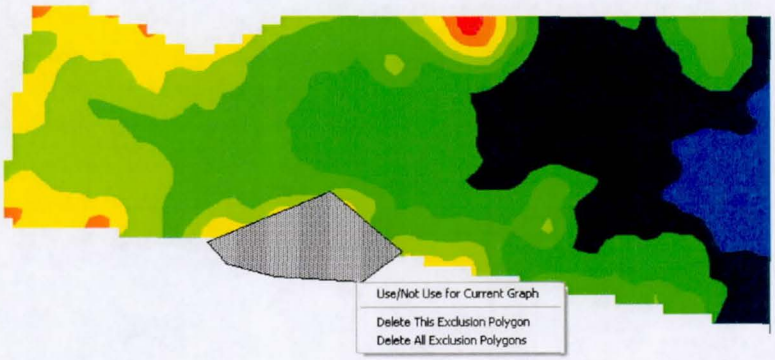
Next use the Apply Exclusion Polygons to Current Graph Command:



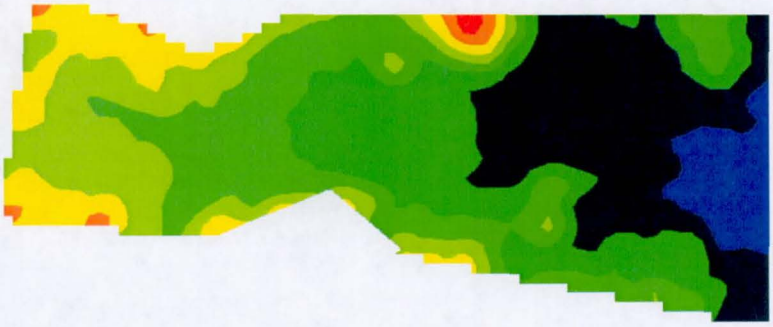
Mapper will erase the plot under the polygon:



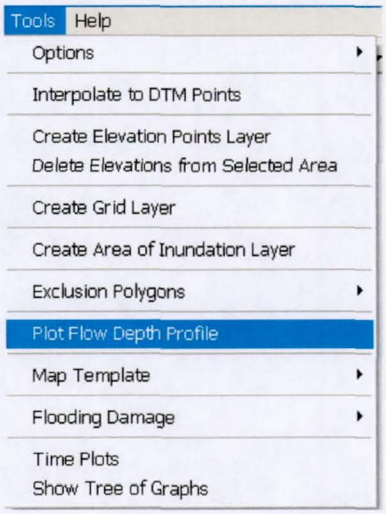
If you click on the polygon border, a shortcut menu will appear:



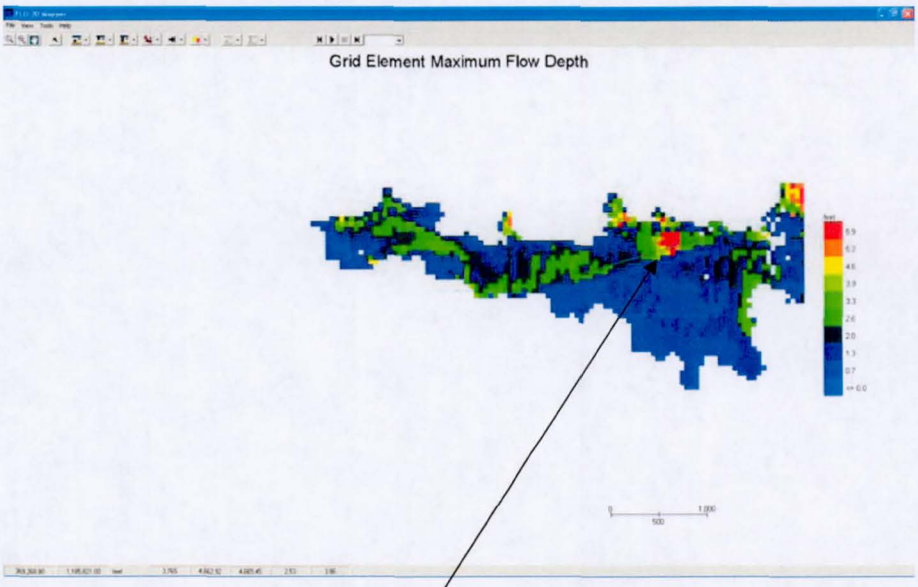
Click on *Delete This Exclusion Polygon* to erase this portion of the contour plot.



Plot Flow Depth Profile Command (Tools Menu)

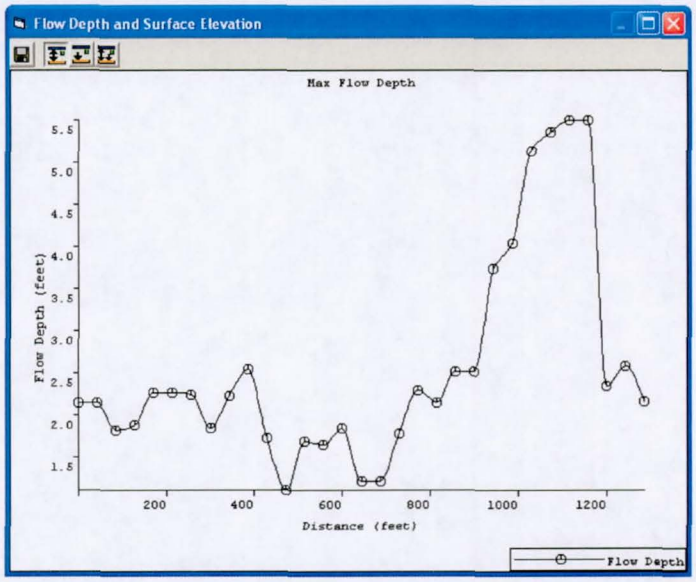


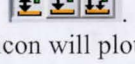
This feature will plot profiles of maximum flow depths, maximum elevations and bed elevation profiles. Click on *Plot Flow Depth Profile* command and draw a line where you want to create the profile. To draw the profile line, click on the first point and locate the mouse cursor on the second point and double click. **The profile line must be within the computational domain.**

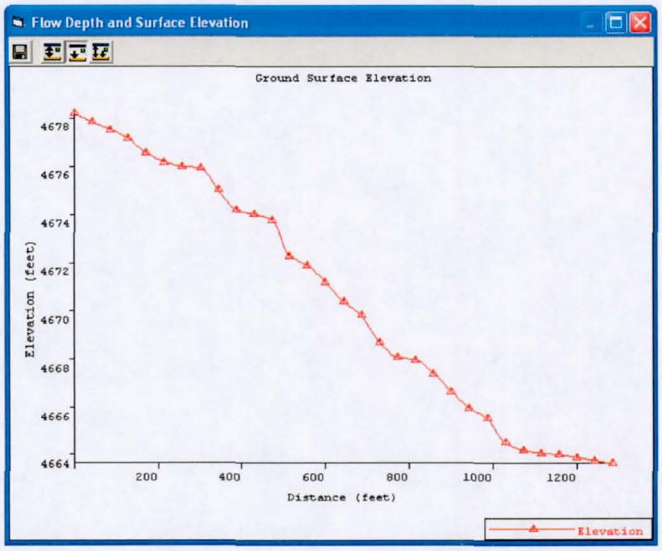


Profile Line is located here:

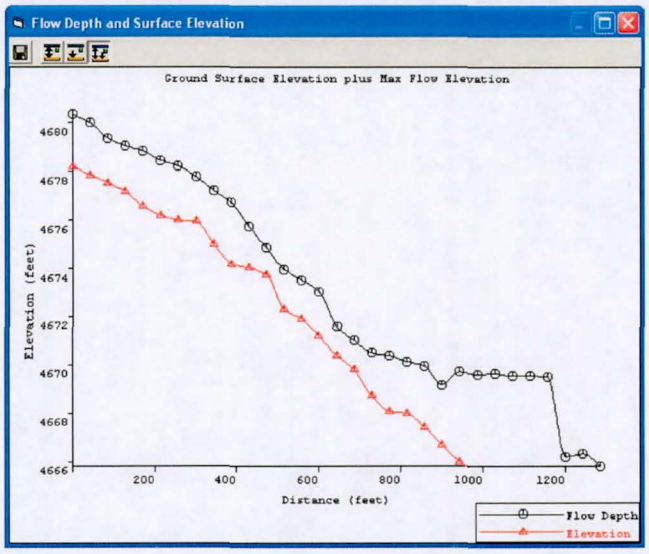
After double clicking on the second point of the line, the following window appears showing the maximum depth profile:



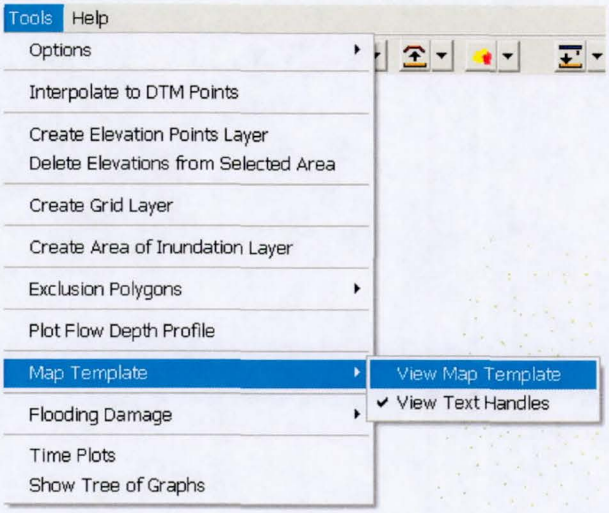
In the upper left corner of the window there are three icons:  . The first icon plots the Maximum Flow Depth profile (shown above). The second icon will plot the Ground Surface Elevation profile:



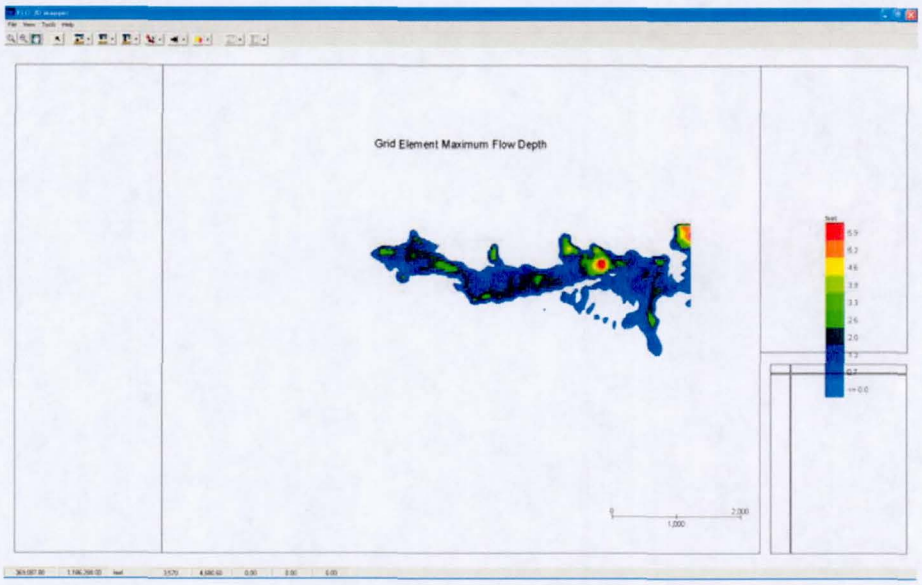
The third icon is used to plot the combined maximum depth and ground elevation profile:



Map/View Map Command (Tools Menu)



This command assists you to create production maps with text. Select the *Map/View Map* option from the *Tools* menu and the system rescales the maximum depth plot and draws a template frame.

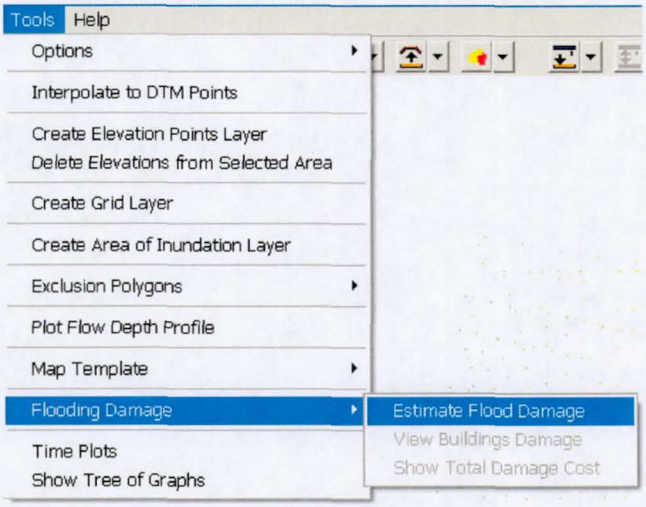


The map template includes a left column for text and a right column with two parts; the upper part for the legend/text and the bottom part for the map ID. The user can edit, move, cut and paste text to both columns. If text already exists, clicking on the text region will change its position. By clicking on the text region with the right mouse button, a pop-up menu will appear with the options: *Delete text*, and *Edit text*. The *Edit text* option enables a dialog box for new text entry or to copy and paste existing text. If there is no text in the text region, the user may enter text. In the lower right region, the user can insert an image that represents a company logo or other map logo. The map legend and scale can be move around. After the map has been completed, it can be exported as one of the supported images for final printing.

Map/View Text Handles Command (Tools Menu)

Use this command to view or hide text handles on the map text and legends. If this option is selected, the text will display a dashed line rectangle around each paragraph. This facilitates locating and moving existing legends.

Flooding Damage/Estimate Flood Damage Command (Tools Menu)



Use this command to estimate the flood damage. It can be used for any type of structure or land use such as agricultural crops. To use this option, you must have the following data:

- A polygon shape file containing an index for each polygon. Each polygon represents a building or house.
- A table file (*.TBL) containing damage ($) data as function of flood depth for each building type in the polygon shape file. This table file is organized as follows:

  Column 1: A damage code coincident with the ID of the shape file polygon.

  Columns 2 to 11: Cost associated with 1 (ft or m) depth, 2 ft depth, …, 10 ft depth.

The following data forms a typical damage table:

```
DamageCode,Depth1,Depth2,Depth3,Depth4,Depth5,Depth6,Depth7,Depth8,Depth9,Depth10
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. The second line 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.

The general procedure used to calculate damage cost is as follows:

1. Read FLO-2D results. This will include maximum flood depth for each grid element.

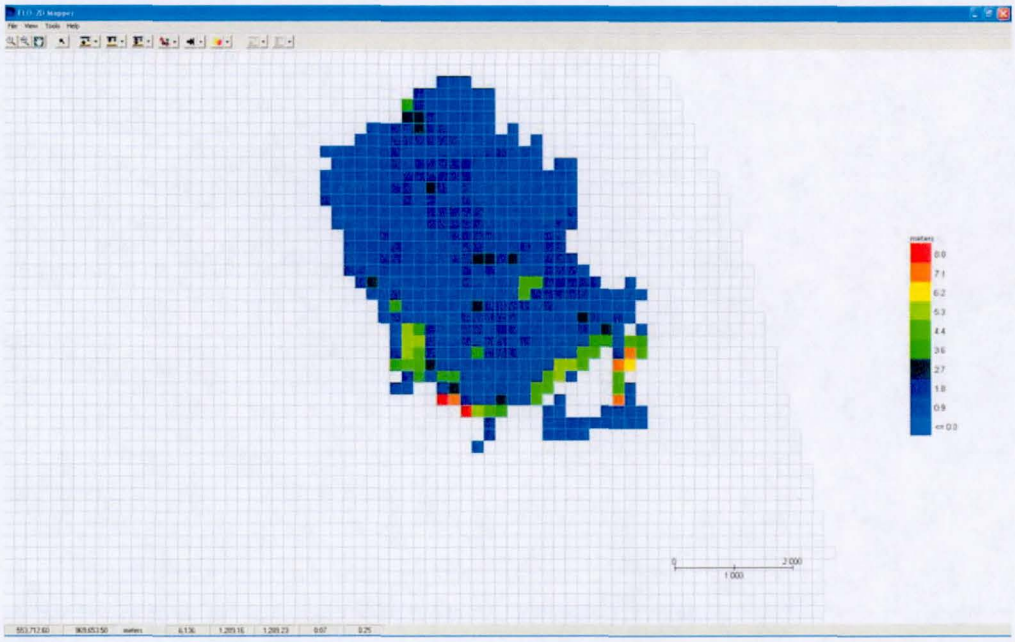
2. Import building polygon shape file.
3. Import cost table ($$ associated with the flow depth for each structure, field, etc.).
4. Determine building (or field) polygon intersections with the FLO-2D grid elements.
5. For the building (or field) polygons, calculate damage $$ using the following formula:

$$COST_B = \left(\frac{\sum A_i (Cost_i)}{A_B} \right)$$

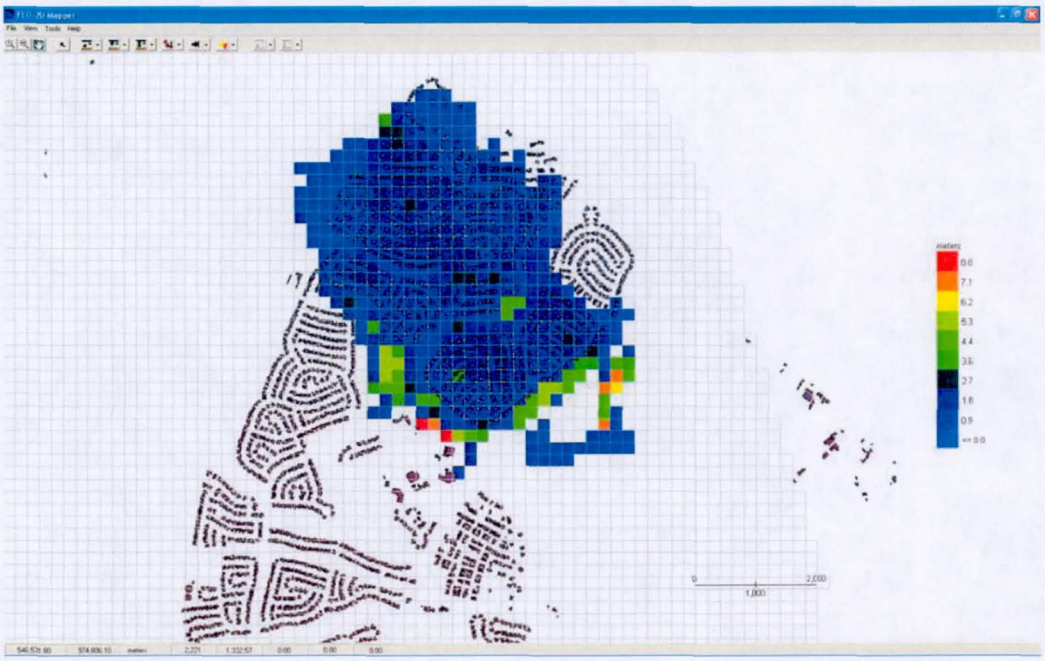
where $COST_B$ is the estimated cost caused by the flooding for building B, A_i is the subarea of intersection between the building and the grid element i. $Cost_i$ is the $$ associated with the area flooded by grid element i, and A_B is the area of building B.

When the algorithm is complete, each building or field will have an estimated flood damage estimate. An example application of this procedure follows. Note that the sample flooding and the cost table are hypothetical:

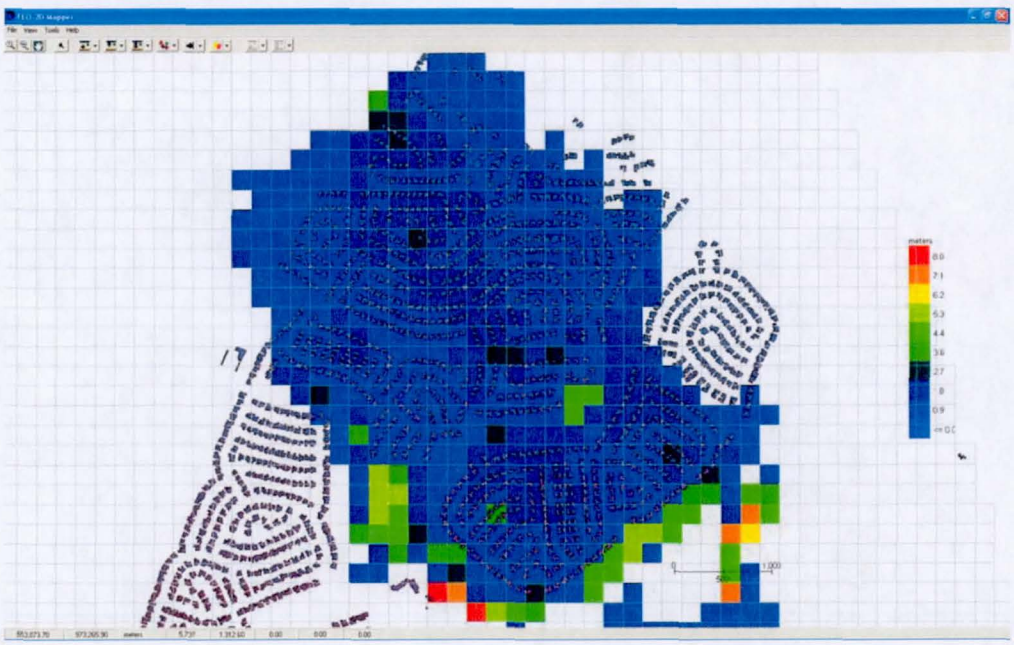
1. Read FLO-2D results. The following figure shows the FLO-2D predicted maximum flood depth:



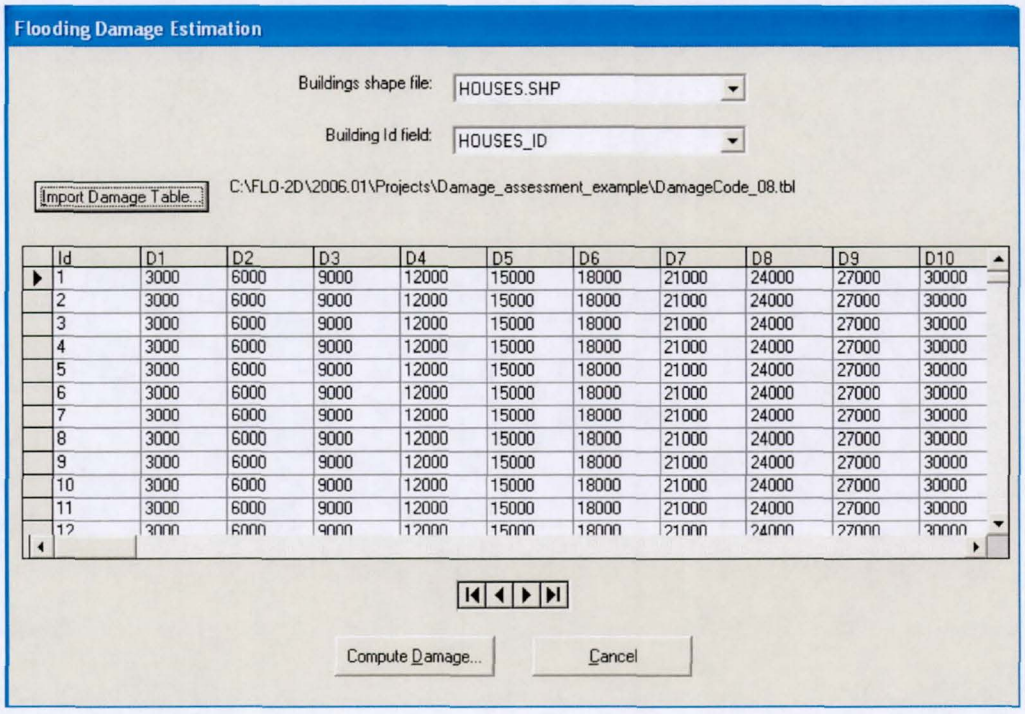
2. Import a building polygon shape file using the *Import Shape file* Command in the *File* menu.



The following figure shows a detail of the building shape file displayed over the maximum flow depth plot:

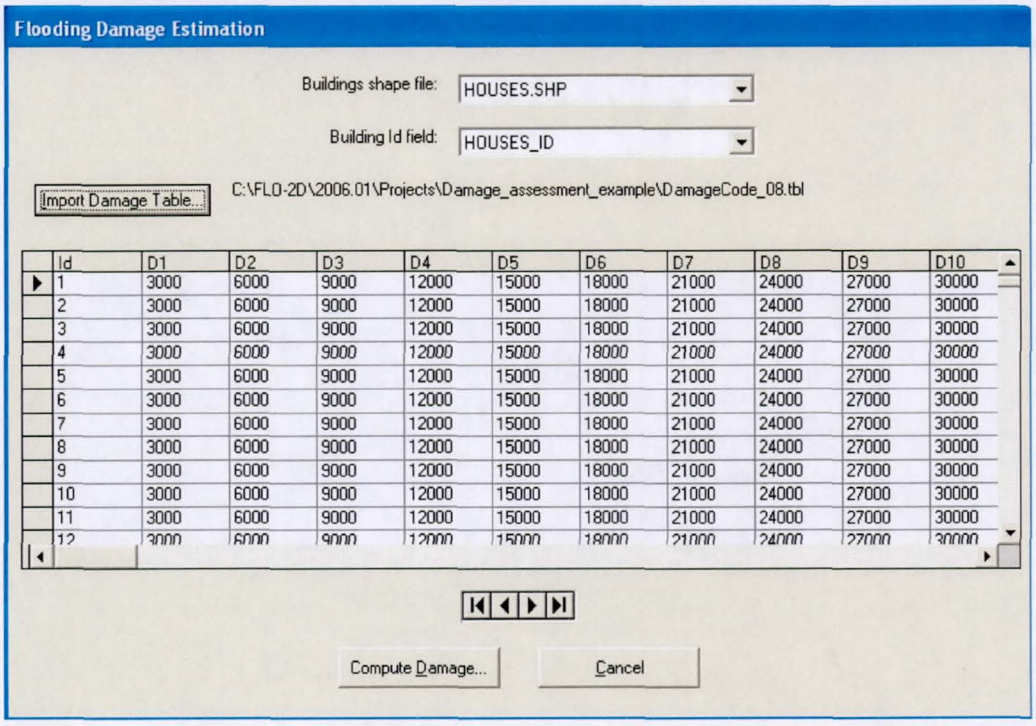


3. Now use the *Estimate Flood Damage* Command in the *Tools* menu. In this example, the buildings shape file is named HOUSES.SHP. The following dialog appears:



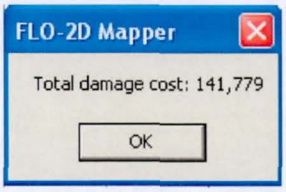
Id	D1	D2	D3	D4	D5	D6	D7	D8	D9	D10
1	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
2	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
3	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
4	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
5	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
6	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
7	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
8	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
9	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
10	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
11	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
12	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000

4. Choose the HOUSES.SHP building shape file from the list and the appropriate building ID field (HOUSES_ID in this case).
5. Import damage table using the button *Import Damage Table*.



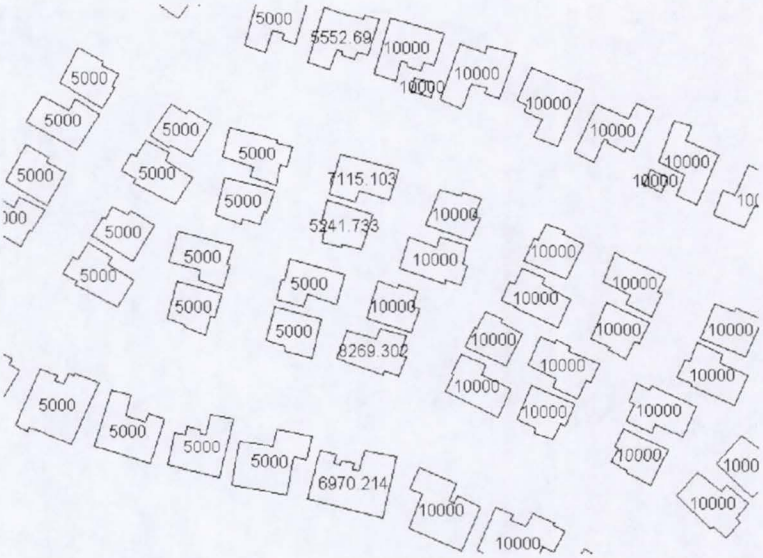
Id	D1	D2	D3	D4	D5	D6	D7	D8	D9	D10
1	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
2	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
3	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
4	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
5	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
6	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
7	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
8	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
9	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
10	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
11	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000
12	3000	6000	9000	12000	15000	18000	21000	24000	27000	30000

6. Click on *Compute Damage* button to calculate damage estimate for each building. *Mapper* will create a layer containing the building cost for each building intercepted by the flooding area. The total damage cost will be displayed:

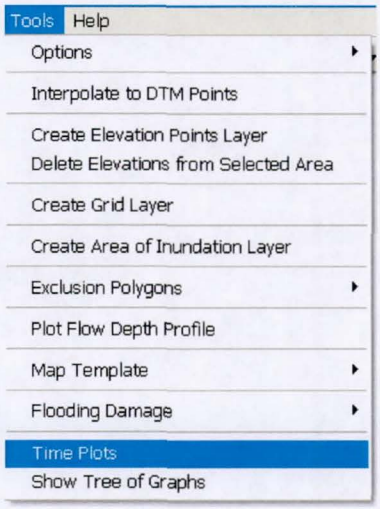


The system also assigns a color for each building according to the damage cost:

The individual building damage cost can be displayed by double clicking the building damage layer in the *'Layers List'*, selecting the *'Adv Label'* option and clicking the *'Apply'* button. It may be necessary to reduce the font size with the *'Label height in map units'* control in the same option. See the following figure.

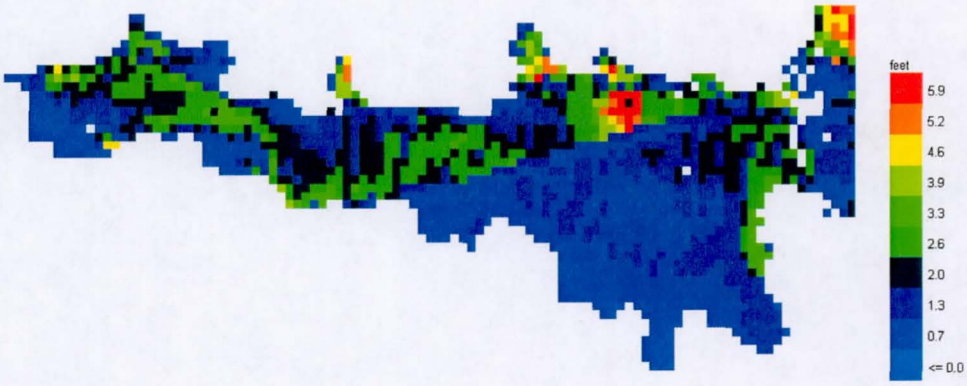

5000
5552.69
10000
10000
10000
10000
10000
10000
10000
10000
5000
5000
5000
5000
5000
5000
7115.103
5000
5241.733
10000
5000
5000
10000
10000
5000
5000
10000
10000
10000
5000
8269.302
10000
10000
10000
10000
10000
10000
10000
10000
5000
5000
5000
5000
6970.214
10000
10000
10000
10000

Time Plots Command (Tools Menu)

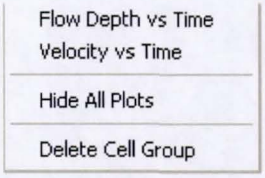


Use this command to select the grid elements and plot flow depth or velocity vs. time.

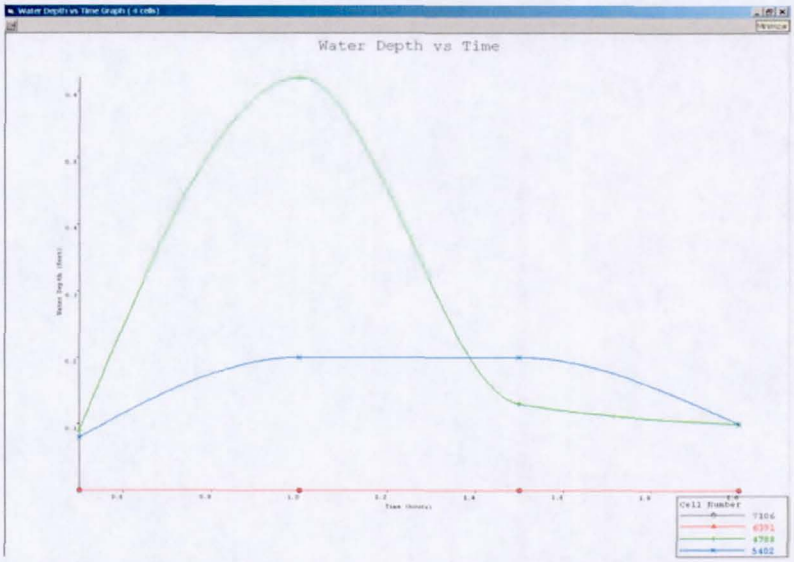
NOTE: This command is only available when the *Time Lapse Output* option was selected in GDS (assign ITIMTEP and TIMTEP variables in the CONT.DAT file for the FLO-2D model simulation). The model will then create the TIMDEP.OUT file that contains the depth and velocity time history by output interval for all grid elements. If this option was not selected before running the model the *Time Plot* Command will be available in the *Mapper* menu.



In the above figure displaying the maximum flow depth, four grid elements were selected. By double clicking on any of the selected elements and the following menu appears:



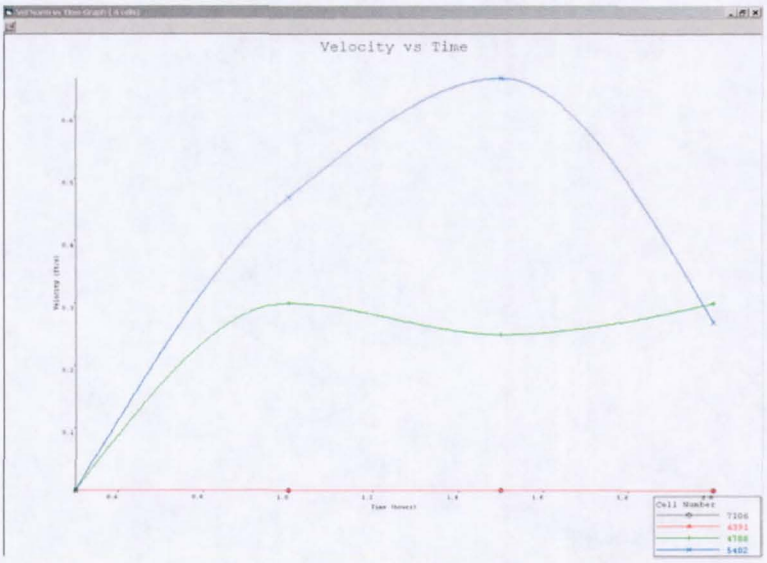
Choose for example the *Flow Depth vs. Time* option. The following graph is displayed:



The legend indicates the grid element number. Only those grid elements with a flow depth greater than zero are plotted.

The other menu option is a plot of *Velocity vs. Time*:

Click on this command to plot the velocity versus time for each selected grid element:



Hide All Plots (Time Plots Shortcut Menu):

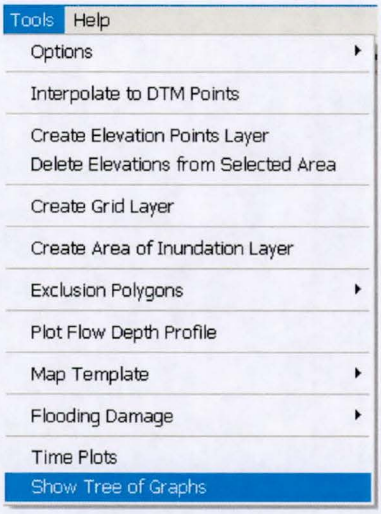
This command hides all plots.

Delete Cell Group (Time Plots Shortcut Menu):

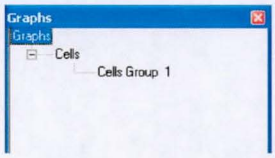
Deletes the selected cell or grid element group

Show Tree of Graphs (Tools Menu):

Displays all the existing plots associated with the grid elements groups.



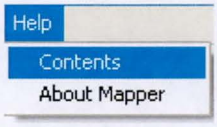
The following window is displays all selected group of Cells or Grid elements.



You may also use this tree to select any available group.

8.4 Help Menu Commands:

Contents Command (Help Menu)



This command initiates the online help system that describes all the *Mapper* functions and menu items. The user may access the help system from any *Mapper* also using the F1 key and a window displaying the corresponding information will be presented.

IX. References

Fiebiger, G., 1997. "Hazard Mapping in Austria." *Journal of Torrent, Avalanche, Landslide and Rockfall Engineering* No.134,Vol.61.

Garcia, R. & Lopez, J.L, 2005. "Debris Flows of December 1999 in Venezuela." *Chapter 20th of Debris-flow Hazards and Related Phenomena*. Jakob, Matthias, Hungr, Oldrich Eds. Springer Verlag Praxis, Berlin.

Garcia, R, J.L. López, M. Noya, M.E. Bello, M.T. Bello, N. González, G. Paredes, M.I. Vivas & J.S. O'Brien, 2003. "Hazard mapping for debris flow events in the alluvial fans of northern Venezuela." *Third International Conference on Debris-Flow Hazards Mitigation: Mechanics, Prediction and Assessment*. Davos, Switzerland. September 10-12.

García, R., López, J.L., Noya, M.E., Bello, M.E., González, N. Paredes, G. & Vivas, M.I., 2002. "Hazard maps for mud and debris flow events in Vargas State and Caracas." *Avila Project Report*. Caracas, Venezuela, (In Spanish).

JICA. Japan International Cooperation Agency, 2000. *Maps of distribution of sediments in Vargas state*. Caracas, Venezuela, 2000.

O'Brien, J.S., Julien, P.Y. & Fullerton, W.T., 1993. "Two-Dimensional Water Flood and Mudflow Simulation," *Journal of Hydraulic Engineering*, ASCE, Vol.119(2).

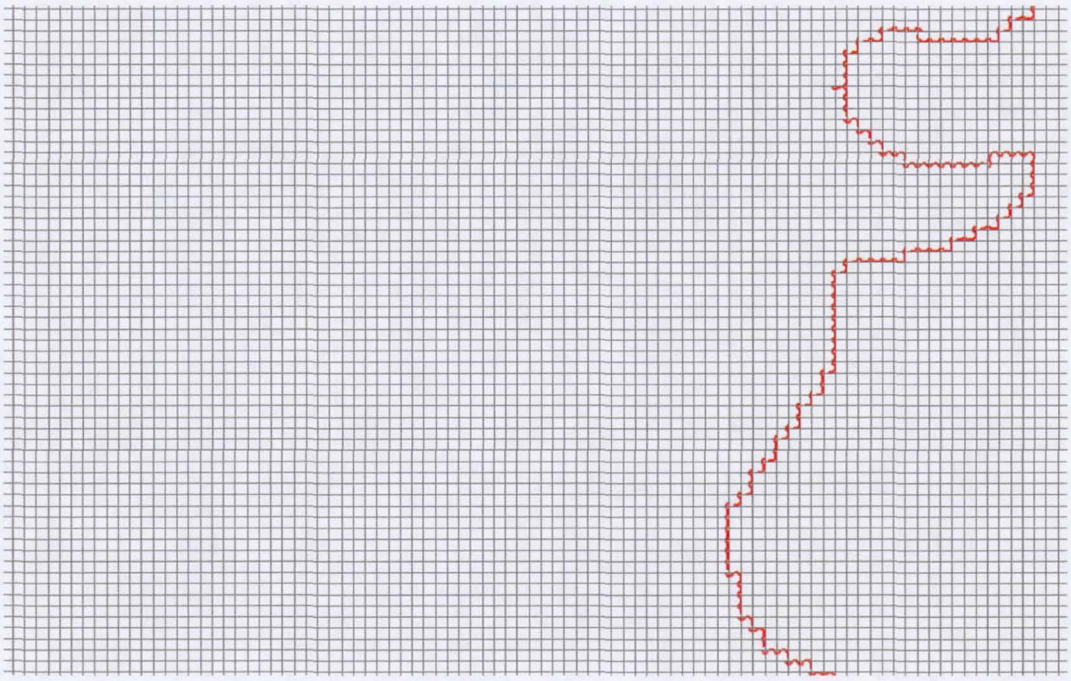
OFEE, OFAT, ODEFP (Switzerland) Ed., 1997. "Prise en compte des dangers dus aux crues dans le cadre des activités de l'aménagement du territoire," Office fédéral de l'économie des aux (OFEE), Office fédéral de l'aménagement du territoire (OFAT), Office fédéral de l'environnent, des forets et du paysage (OFEFP), Bienne.

PREVENE. 2001. *Contribution to "Natural" Disaster Prevention in Venezuela*. Cooperation: Venezuela - Switzerland - PNUD (Project VEN/00/005).

FLO-2D

GRID DEVELOPER SYSTEM

INTERPOLATION METHOD FROM MULTIPLE FILES



FLO-2D Software, Inc.

GDS new interpolation tool

Previous versions of the FLO-2D Grid Developer System (GDS) provided a method to interpolate terrain elevations that required importing all points. The original procedure was not able to handle more than about 15 million points due to limitations on the development tools used to create GDS. To circumvent this limitation and allow the processing of virtually unlimited number of elevation point data, FLO-2D Software has developed a new method that does not require importing the points and significantly extends the model interpolation capability. This document summarizes the functionality of the new tool and provides simple instructions how to use it.

Data requirements

The new interpolation tool reads data files in any of the following data ASCII formats.

1) Space or comma delimited ASCII files, 3 data values per line

   These files contain three values on each line as follows:
   X_Coordinate Y_Coordinate Elevation
   Example of space delimited file:

```
369941.27   1183607.125   4654.71
369946.27   1183608.125   4654.50
369951.27   1183609.125   4654.29
```

   Example of comma delimited file:

```
369941.27 ,  1183607.125 ,  4654.71
369946.27 ,  1183608.125 ,  4654.50
369951.27 ,  1183609.125 ,  4654.29
```

2) Space or comma delimited ASCII files, 5 data values per line

   These files contain five values on each line as follows:

ID X_Coordinate Y_Coordinate Elevation Value

For this file format only the point coordinates and elevation value is used. First column (ID) and last column (Value) are ignored.

Example of space delimited file:

```
X1   369941.27   1183607.125   4654.71 777
X2   369946.27   1183608.125   4654.50 71717
X3   369951.27   1183609.125   4654.29 2828
```

Example of comma delimited file:

```
X1 , 369941.27 , 1183607.125 ,  4654.71 , 777
X2 , 369946.27 , 1183608.125 ,  4654.50 , 71717
X3 , 369951.27 , 1183609.125 , 4654.29 ,  2828
```

3) List of elevation data files ELEVFILES.DAT:
   This optional file contains the list of files including path and name. It needs to be created by the user with any text editor program and facilitates handling hundreds or thousands of files which may be difficult to select using a standard MS-Windows file handling dialog box. Users may mix any of the supported formats, i.e., some files may be 3 column space delimited, and others five columns comma delimited, etc. All files must have.TXT extension.

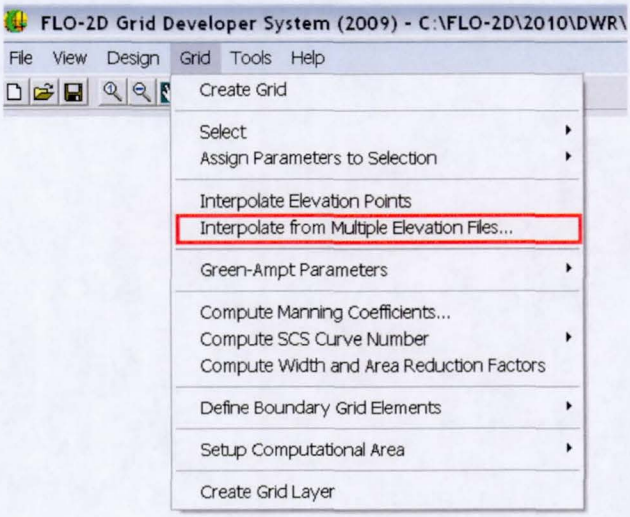
   Example of ELEVFILES.DAT file:

```
C:\PROJECTS\FLO2D\2009\Goat2009\dtm1.TXT
C:\PROJECTS\FLO2D\2009\Goat2009\dtm2.TXT
C:\PROJECTS\FLO2D\2009\Goat2009\dtm5col.TXT
C:\PROJECTS\FLO2D\2009\Goat2009\dtm5col2.TXT
```

***NOTE**: All elevation ASCII files, regardless of their format must have extension .TXT.*

User instructions

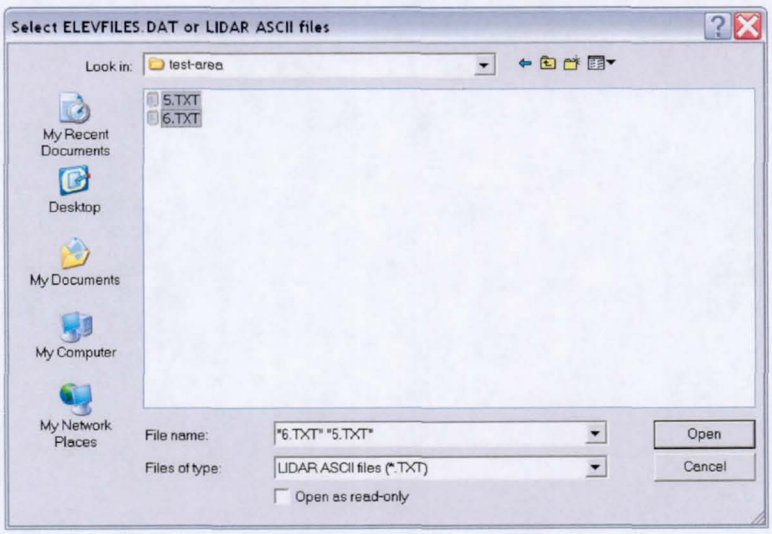
GDS requires that the user creates a FLO-2D grid prior to interpolation. To interpolate use the new *Interpolate from Multiple Elevation Files...* from the GDS Grid menu as shown:



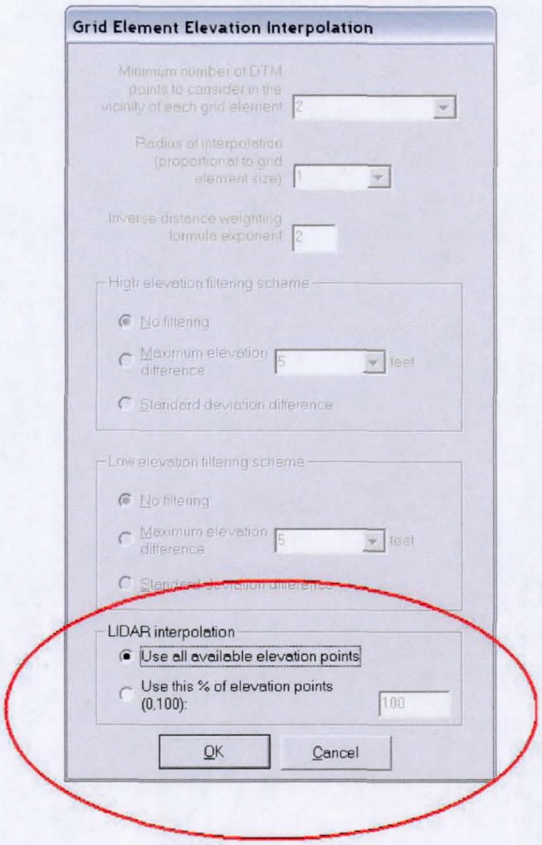
This command will bring up a file selection dialog:

There are two options here:

- Users can select an ELEVFILES.DAT file if available , and import the list of elevation files in the format discussed in the previous section or,
- Users can select the actual elevation data files. It is necessary to change the default file extension to .TXT to select a group of ASCII elevation files as follows:



For either option, clicking *Open*, displays the interpolation dialog box:

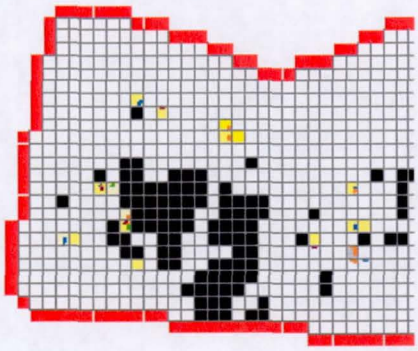


There are two interpolation option to chose. The default is *Use all available elevation points,* which will compute the interpolated elevations using all points on each grid element. Selecting the *Use this % of elevation points* will calculate the elevation using the assigned subset of points on each grid element. The % is a global value based on the average number of elevation points per grid element.

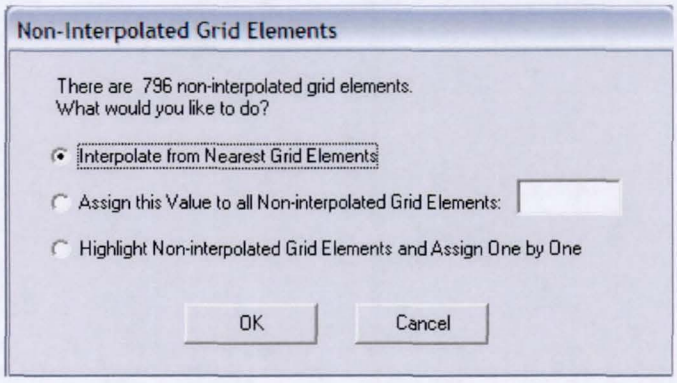
Interpolation method

If the user selects *Use all available elevation points* option, the GDS will compute the grid element elevation using the elevation average of all points on the element. The GDS does not import or display the elevation points, but instead reads one point at a time, determines the grid element where the point is located and computes the average elevation. A similar procedure is followed if the user selects a subset of points but GDS will calculate the elevation using the indicated percentage of points on the element.

If no elevation points are found on an element or if the number of points is less than the specified percentage, a tag value of -9999 will be assigned to that element. Also the element will be highlighted in black as shown:



If there is at least one element without a calculated elevation, a dialog box will be displayed presenting three options to assign elevations to those grid elements:

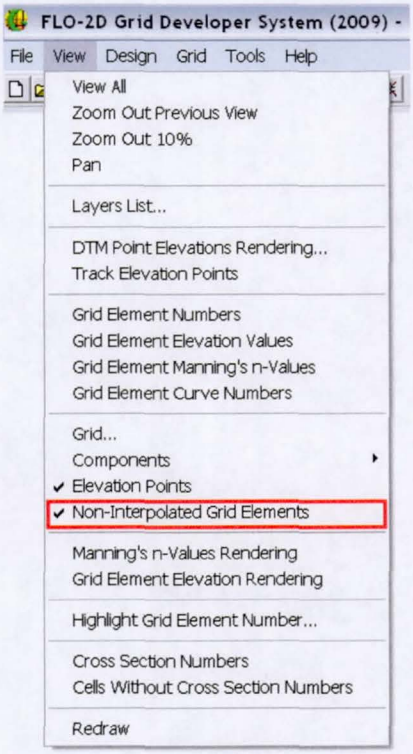


The *Interpolate from Nearest Grid Elements* option will

compute an elevation by assessing the elevation data in the neighboring 8 grid elements and averaging the point elevations (-9999 values are not used). The algorithm initiates from the periphery of a cluster of non-interpolated grid elements and proceeds to the interior of the cluster ensuring that at the end of the procedure all grid elements will be assigned an elevation.

The user can assign a value to all -9999 grid elements using *Assign this Value to all Non-interpolated Grid Elements.* The third option will leave all uninterpolated elements with the -9999 tag. The user will need to double click and assign a desired elevation for each one.

The new *Non-Interpolated Grid Elements* command on the *View* menu turns on or off the black unassigned grid elements.



***NOTE**: GDS will not initiate a FLO-2D model if there are elements with the -9999 tag.*

Support

GDS support questions or comments may be addressed by email to reinaldo@flo-2d.com or jim@flo-2d.com.

Welcome to the FLO-2D DFIRM Tools

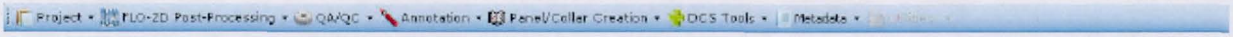
These are a series of tools designed to help with DFIRM creation. There are two main working areas: the main program, and two toolbars that function inside of ArcView. Both work with the same project file created inside the main program. You start by creating a project for your community.

They are not a substitute for more advanced tools that have been created. The goal is to create a set of flexible and dynamic tools to automate many repetitive tasks. They are flexible in that the user maintains control over nearly every output product. They are dynamic in that they retrieve a lot of information from two easily editable xml files saved on the computer. It is not recommended to edit these, but they are available for the user to change. The user should review the final products created by the tools for problems that might have arrised.

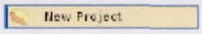
Project Introduction

To work with the DFIRM tools, you need to create a new project. Some tools on the toolbars do not require the project to be loaded. Most tools in the main program will require the project to be loaded, and are disabled till a project has been created or opened. The project keeps track of where the original files are located, but does not check if they exist till they are needed. The project file also stores metadata information, and uses this when creating DFIRM metadata. The project file is just an xml file in the project directory. While it is possible to manually edit this file, it is not recommended.

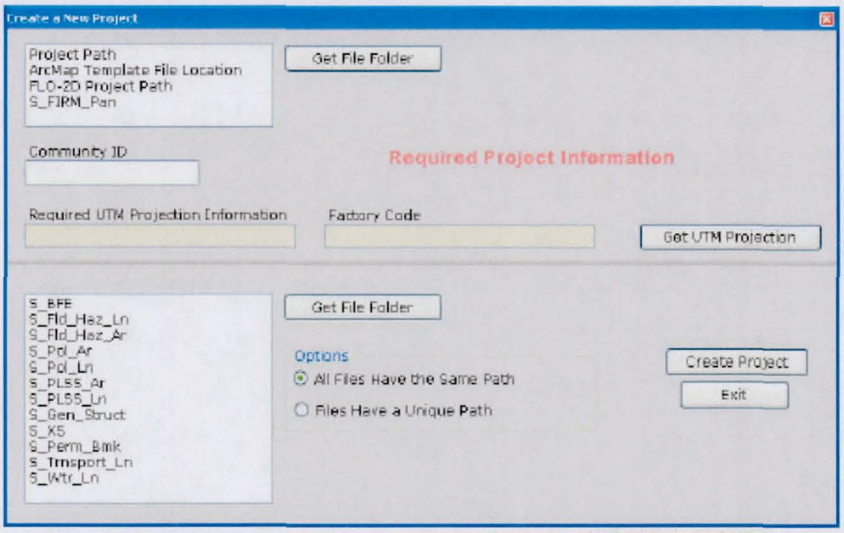
Create a new project



- Select New Project from the Project dropdown menu.

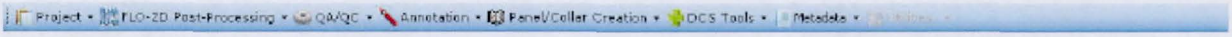


- This opens the New Project dialog.

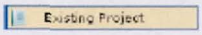


- First select the Project Path in the textbox. Then click Get File Folder and navigate to where you would like to create a project folder. The project folder will be creat selected folder.
- Select ArcMap Template File Location and navigate to where you would like to store the Panel Template. It is recommended to store this in a location that is accessib working on the project, otherwise it will need to be copied from computer to computer.
- Repeat for the FLO -2D Project Path. If you do not have a FLO-2D project path, select an arbitrary folder.
- For the S_FIRM_Pan option, you need to select the folder where a S_FIRM_Pan shapefile is located. The file does not need to be there at the time of project creation, tools.
- Enter a Community ID
- Click the Get UTM Projection button and you should be taken to ESRI's Coordinate Systems Folder. Select a UTM projection for your community. While FEMA doc Coordinate Systems, UTM is the preferred coordinate system.
- Next select S_BFE. If all your DFIRM shapefiles are stored in the same location, leave the default option selected. You do not need to assign a path to each one.
- Click Create Project.
- If this is a project you plan to use next time. Click on File in the main window, and save settings.

Open an existing project

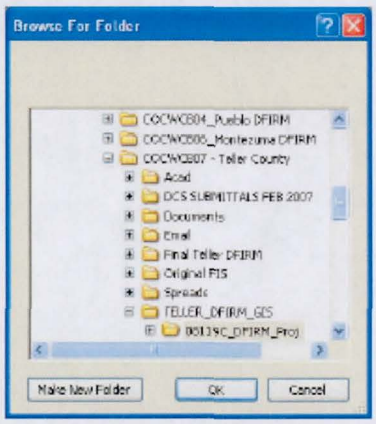


- Select Existing Project from the Project dropdown menu.



- This opens the Open Existing Project dialog.

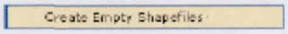
- Click the Get Project Folder button



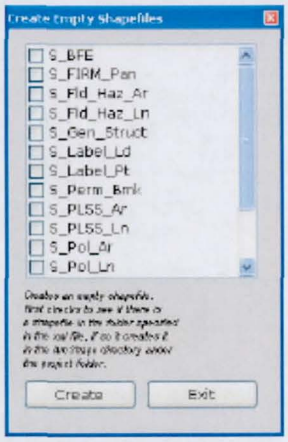
- Navigate to the project folder that you have created. It should end in DFIRM_PROJ. Select this folder and click ok.
- In the Open Existing Project dialog, the Community ID textbox should be populated with the project's Community ID.
- Click on Get Project.
- If this is a project you plan on using next time. Go to File in the main window, and select Save Settings.

Create empty shapefiles

- Select Create Empty Shapefiles from the QA/QC dropdown menu.

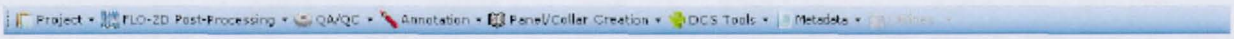


- This opens the Create Empty Shapefiles dialog.

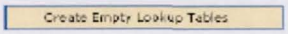


- Check which empty shapefile you would like to create.
- Click Create.
- If the file does not exist in the stored project path for that file, then the file will be created there. Otherwise it will be created in the project directory under MSC Deliverables\communityID_DFIRM\DFIRM_DB\ArcShape.
- Tip: If you are working with a geodatabase, have the DFIRM tools create empty shapefiles for your feature classes. Then use the Append Tool in the ArcMap/ArcCat your geodatabase feature class to the new shapefile. Exporting to a shapefile from a feature class creates incorrect field widths, appending maintains the correct structu names match.
- Tip: If there is a shapefile that is not listed that you need to create, you can add it to the dfirm.xml file located in the install directory/Supportfiles. Be sure to add a pat the new shapefile.

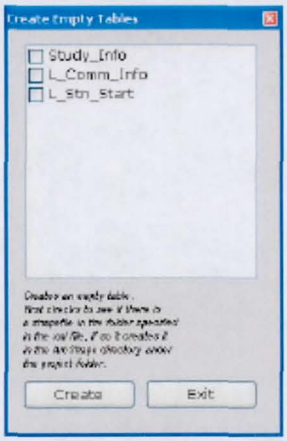
Create empty lookup tables



- Select Create Empty Lookup Tables from the QA/QC dropdown menu.



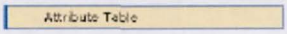
- This opens the Create Empty Lookup Tables dialog.



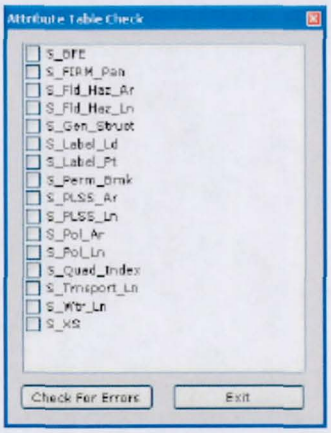
- Check which empty table you would like to create.
- Click Create.
- The file will be created in the project directory under MSC Deliverables\communityID_DFIRM\DFIRM_DB\ArcShape.
- Tip: If there is a table that is not listed that you need to create, you can add it to the dfirm.xml file located in the install directory/Supportfiles. Be sure to add a path in new shapefile.

QA/QC the attribute tables

- Select Attribute Table from the QA/QC dropdown menu.



- This opens the Attribute Table dialog.



- Check which dfirm table you would like to check.
- Click Check for errors.
- An error xml file is created in the project folder.
- Tip: If there is a shapefile that is not listed that you need to create, you can add it to the dfirm.xml file located in the install directory/Supportfiles.

Check vector topology

- Select Topology from the QA/QC dropdown menu.

- This opens the Topology Check dialog.

- The topology check works with either points or polylines.
- The Point section primarily looks for for duplicate points. Polyline checks for undershoots and overshoots.
- For the polyline check, you need to enter a search radius. Basically an acceptable distance where overshoots and undershoots might occur.
- The output is in the main window. A point shapefile of errors is also created.
- Note: This is not a substitute for more advanced topology checks. It may not catch every error.

Annotation

Creating annotation works in two parts. The pre-processing tools are in the main DFIRM tools window, and the creation tools are located on the DFIRM Tools Toolbar; this was done to speed up the process. The pre-processor takes the existing shapefiles and imports them into a geodatabase. While doing this, preprocessor attempts to categorize and classify each feature to identify what styles the annotation will use. It also attempts to create the text string based on certain fields. That is why it is necessary to have complete feature classes when creating annotation. There is also an option to replace or add feature classes to the geodatabase. At this point, the user should review the created feature classes to check that the pre-processing worked. It is easy to change these classifications and text strings at this point, because they are stored as attributes in the file.

On the DFIRM toolbar, there is an option to create annotation or to setup annotation for editing. If you prefer to generate your own annotation from labels, select the setup annotation for editing option. This associates label styles stored in the DFIRMv2.style file with the layers and creates label classes for these files. You can view these in the label manager. Otherwise, run the create annotation tool. This generates a new geodatabase with the annotation. You can create annotation for all the panels or individual panels. The S_Pol_Ln file will not create annotation, but you can create an empty annotation class at each of the three scales. The political boundary labels will need to be created manually. At this point there is now automated tool. The S_Pol_Ar does work fine.

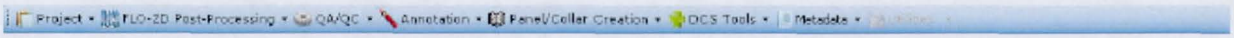
Now you have all the annotation stored in a geodatabase that can be easily moved and edited (en masse or individually). Manually editing is the best method to ensure complete control and quality.

Annotation pre-processing

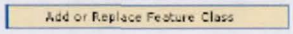
- Select Annotation Pre-Processing from the Annotation dropdown menu.

- There is no dialog for this tool.
- The tool will process automatically creating a geodatabase in the project geodatabase folder.
- Please verify the files after creation.

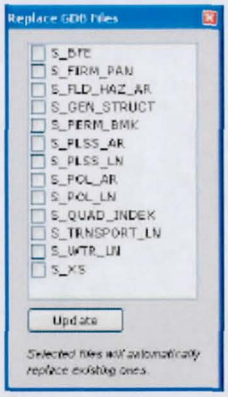
Update Geodatabase



- Select Add or Replace Feature Class from the Annotation dropdown menu.



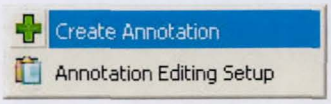
- This opens the Add or Replace dialog.



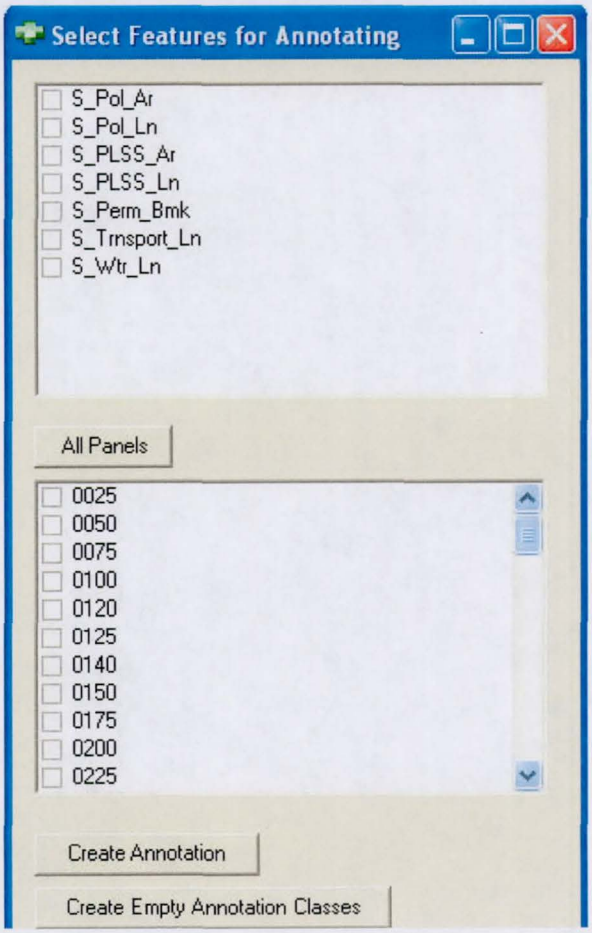
- Simply select which files to add or replace, and the pre-processing will run again.

Automatically generate annotation

- Select Create Annotation from the Annotation dropdown menu.



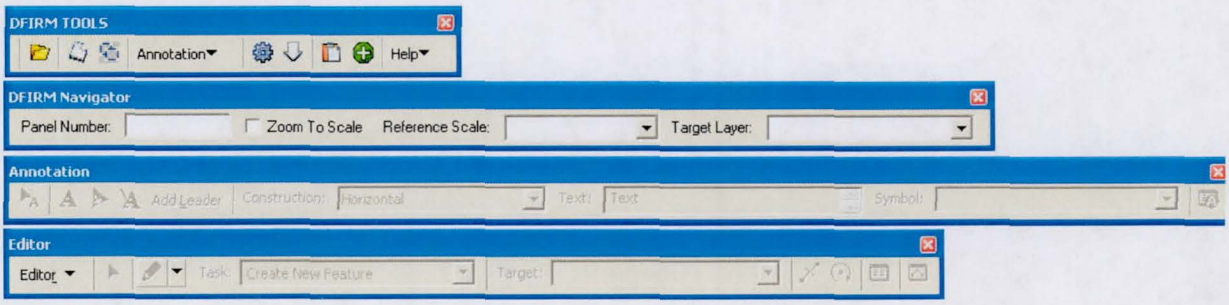
- This opens the Create Annotation dialog.



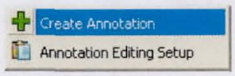
- This dialog requires that a project is open on the toolbar.
- Select the feature class you would like to create annotation for (multiple's are allowed). Do not select the S_Pol_Ln feature class.
- Select the panel.
- Click on Create Annotation.
- If you want to create a S_Pol_Ln annotation class, select it and click Create Empty Annotation Classes

Editing annotation in ArcMap

Editing the annotation will involve at least four toolbars: the DFIRM TOOLS, DFIRM NAVIGATOR, ANNOTATION, and EDITOR:



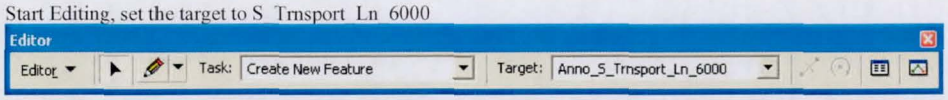
- Select Create Annotation from the Annotation dropdown menu.



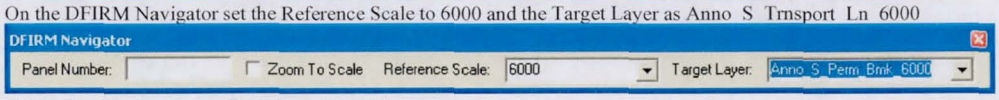
- Stylize the base feature classes to your liking (roads = red, streams = blue, etc)
- Add the editor toolbar. Click the editor drop down, and select Start Editing. Select the Annotation Geodatabase when asked, and then set the Target as the annotation you like to edit. Through these examples we will be using the S_Trnsport annotation. You will notice that the prefix always matches the spatial database layer, and is followed by the scale. The scale of the annotation is dependent on that panel's scale.
- If you would like, you could generate the panels and then edit the annotation from within the panel, but for this example, we will edit the whole community at the same time (in this case a county). If you are editing the whole community, there is a tool on the DFIRM TOOLS Toolbar that will generate a shapefile of points separated by 5000 or 1000 spaces. This will line up with the grid tick marks on the final panel. This way you can edit annotation and avoid having the annotation intersect the tick marks.
- The DFIRM Navigator contains tools to assist with Annotation editing. You can type the 4 digit panel number and you will be zoomed to it. If you check the zoom to scale, and enter the panel number, you will be zoomed to that panel and the scale will be set at the same as the panel's scale. The reference scale will set your symbology at the correct scale. Use the dropdown to select which scale you need. The Target Layer sets the Active Annotation Target. You would need to set this each time you want to copy a piece of annotation. For example, you copy a 12000 scale transportation annotation, set it to S_Trnsport_Ln_12000 copy and paste; then you want to copy the 12000 stream, set it to S_Wtr_Ln_12000 and copy paste. You should also set the target layer on the editor toolbar, just to be sure that the annotation is stored in the right feature class. You can manually set the Active Annotation Target by adding the Draw toolbar, click the Drawing menu and select the Active Annotation Target.
- You have three options to select the annotation. The Select Elements tool (Tools Toolbar), Edit Tool (Editor Toolbar), and Edit Annotation Tool (Annotation Toolbar). It is best to select the annotation using the Edit Annotation Tool.

Example:

- Start Editing, set the target to S_Trnsport_Ln_6000

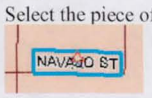


- On the DFIRM Navigator set the Reference Scale to 6000 and the Target Layer as Anno_S_Trnsport_Ln_6000



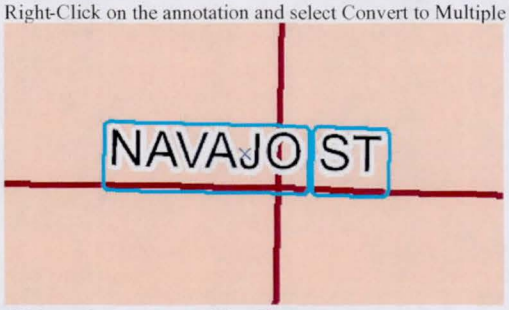
- Select the Edit Annotation Tool from the toolbar.

- Select the piece of annotation you need to edit. A cyan border will surround it.

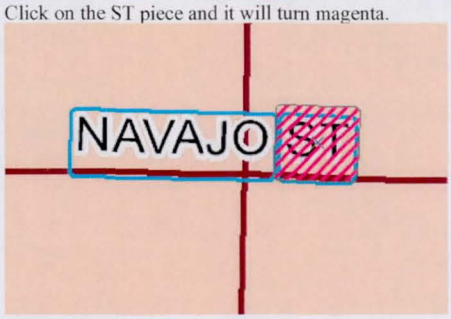


- Now you can move this piece of annotation.
- Right Click and select Rotate Mode. This is a free rotate mode. Click and hold the left mouse button and move the mouse to rotate.

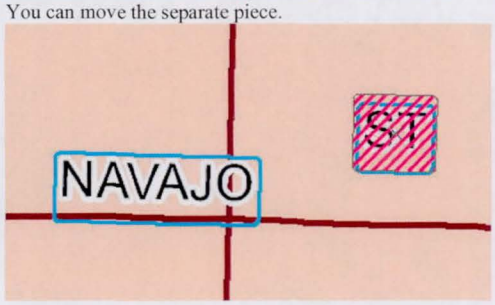
- Right-Click and select Rotate Mode. This is a free rotate mode. Click and hold the left mouse button and move the mouse to rotate.
- A handy feature for annotation is to convert it to multiple parts. This splits the annotation (based on spacing) but maintains it as a single piece.
- Right-Click on the annotation and select Convert to Multiple Parts. The annotation is split into different pieces



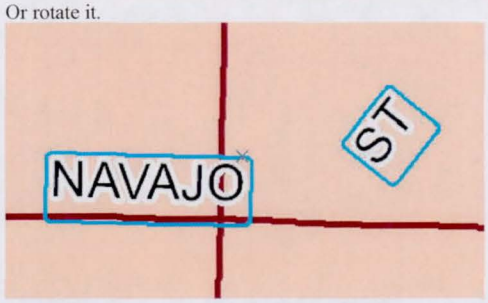
- Click on the ST piece and it will turn magenta.



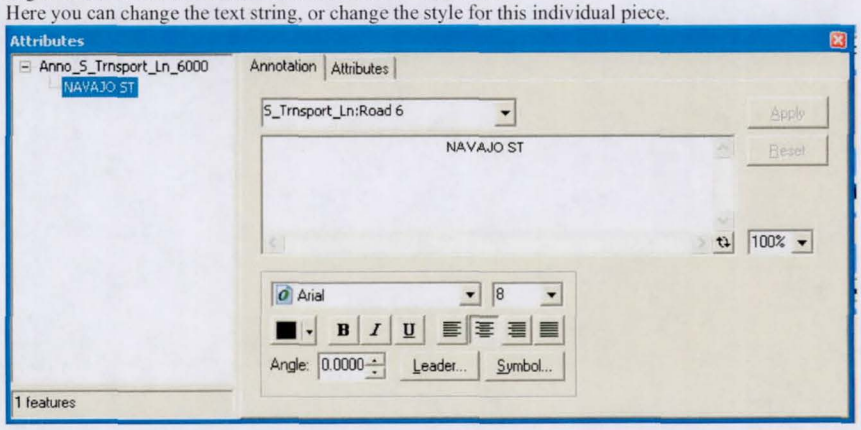
- You can move the separate piece.



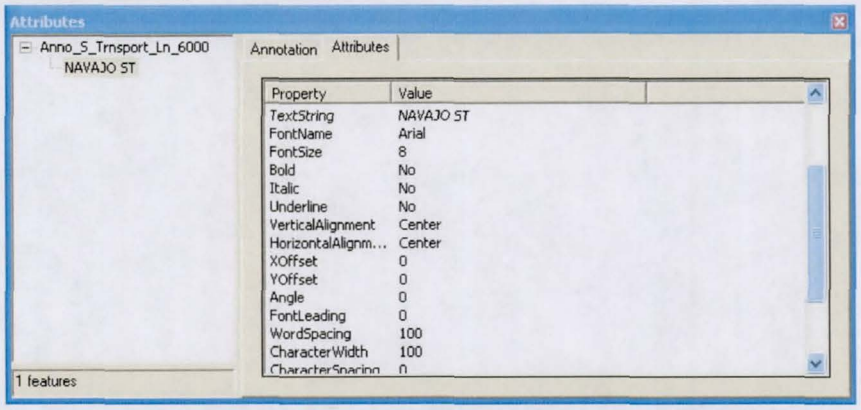
- Or rotate it.



- Right-Click on the selected annotation and select Attributes.
- Here you can change the text string, or change the style for this individual piece.



- On the attributes tab, you can edit the specific attributes:



- For example you can enter a value for a rotation instead of a freehand rotation (the angle field)
- You can also change attributes for the annotation en masse by using the annotations Attribute Table and the Field Calculator

Attributes of Anno_S_Trnsport_Ln_6000

Status	TextString	FontName	FontSize	Bold	Italic	Underline	VerticalAlignment	HorizontalAlignment	XOffset
Placed	6TH ST	Arial	8	No	No	No	Center	Center	0
Placed	10TH ST	Arial	8	No	No	No	Center	Center	0
Placed	CR 411	Arial	8	No	No	No	Center	Center	0
Placed	5TH ST	Arial	8	No	No	No	Center	Center	0
Placed	LEWIS ST	Arial	8	No	No	No	Center	Center	0
Placed	7TH ST	Arial	8	No	No	No	Center	Center	0
Placed	PAGOSA ST	Arial	8	No	No	No	Center	Center	0
Placed	CR 200	Arial	8	No	No	No	Center	Center	0
Placed	CR 400	Arial	8	No	No	No	Center	Center	0
Placed	160	Arial Narrow	8	Yes	No	No	Center	Center	0
Placed	APACHE ST	Arial	8	No	No	No	Center	Center	0
Placed	CR 500	Arial	8	No	No	No	Center	Center	0
Placed	9TH ST	Arial	8	No	No	No	Center	Center	0
Placed	3RD ST	Arial	8	No	No	No	Center	Center	0
Placed	2ND ST	Arial	8	No	No	No	Center	Center	0
Placed	SQUAW CANYON PL	Arial	8	No	No	No	Center	Center	0
Placed	CR 119	Arial	8	No	No	No	Center	Center	0
Placed	HOT SPRINGS BLVD	Arial	8	No	No	No	Center	Center	0
Placed	SAN JUAN ST	Arial	8	No	No	No	Center	Center	0
Placed	4TH ST	Arial	8	No	No	No	Center	Center	0
Placed	CRESTVIEW DR	Arial	8	No	No	No	Center	Center	0
Placed	14TH ST	Arial	8	No	No	No	Center	Center	0
Placed	1ST ST	Arial	8	No	No	No	Center	Center	0

- There are two ways to add annotation: copy existing annotation and paste, or use the Construct Horizontal Annotation tool.
- **A** and set the symbol on the dropdown Symbol: S_Trnsport_Ln:Abandoned Railroa
- By selecting the symbol, the style will be set and you just need to change the text string.
- To copy annotation use the Editor Toolbar's Edit Tool. Set the targets, before copying. Select the annotation, right-click and click copy. Right-click again and select paste.
- To Delete annotation, select the annotation with any of the selection tools, and press the delete key.

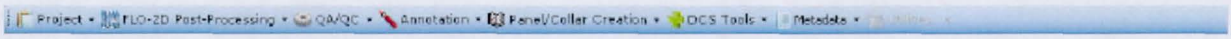
Working with panels

Like with the annotation, the panel generation is broken up into three phases. A setup phase, creation, and editing phase. Whenever a project is created, a template is copied to the template folder the user selects. You start by editing this template. Areas like the notes to users section will need to be changed. To assist in this, there are certain pieces of bold text enclosed in brackets, and these should be changed. You also need to change the projection information and change, or edit the grid/graticule settings. The index map is included, but is not required by FEMA. If you have special template you are using, you can easily replace the main template in the SupportFiles directory. There is also an index map template in the folder, but nothing is done with this; it is up to the user. Blank areas in the titleblock will be populated by the tools.

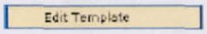
Next you generate the panels. This will create all or individual panels based on what is stored in the S_FIRM_Pan that was created during the annotation preprocessing. Extras include the index map, and adding community information. The community information requires a completed S_Pol_Ar file in the geodatabase. This process creates mxd files for each panel in the Panels folder under the project folder. Each is broken up by scale.

Finally open the panels in ArcMap, and using the Add Grid/Graticule tool, you can convert the grid/graticule to the proper format. If you need to add a Township and Range note (see FEMA's text placement guide), use the add township and range button.

Edit the project's template



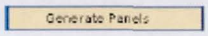
- Select Edit Template from the Panel/Collar dropdown menu.



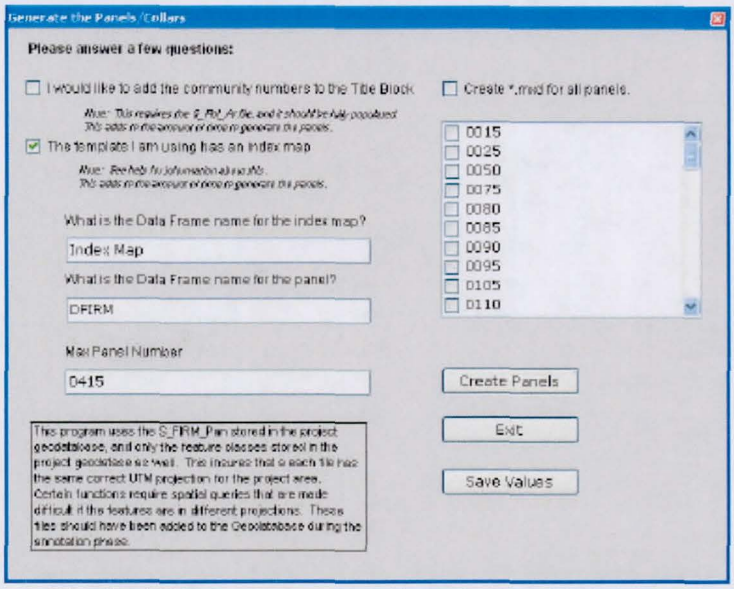
- There is no dialog for this tool.
- The tool will open a session of ArcMap where you can edit the template for your project.
- See the tips and tricks section on how you can easily automate updating your different notes sections.

Create panels/collars

- Select Generate Panels from the Panel/Collar dropdown menu.



- This opens the Generate Panels dialog.

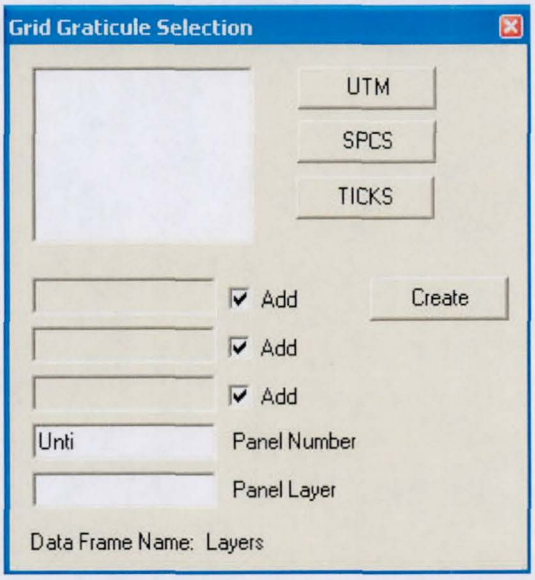


- Select all, or individual panels to generate. At this point there is no distinction between printed and not printed panels
- Select if you would like to add community information.
- Select if you would like to add an index map.
- If you changed the Data Frames name, put that information in the text boxes.
- The max panel number is added based on the last panel on the checked list. Change this if so desired.
- Click Create Panels.
- *It is recommended to start with 2 panels and then verify that everything looks ok, then run the tool overnight.*

Inserting a grid/graticule

- Select the add Grid button.

- This opens the Create Grid dialog.

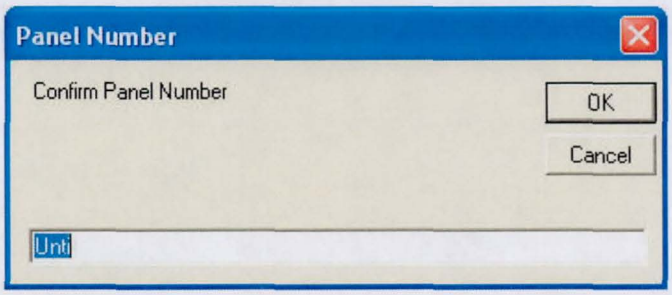


- Select a grid name in the textbox and then corresponding coordinate system for the grids you need to add.
- Confirm Panel number and the S_FIRM_Pan Layers
- Click on Create. The grid should now be verified. Sometimes there will be tick marks that fall outside the border that will need to be cleaned. This is mostly depended on the rotation caused by the UTM projection.

Add Township and Range note

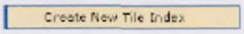
- Select the add Township and Range Note button.

- This opens an input box.

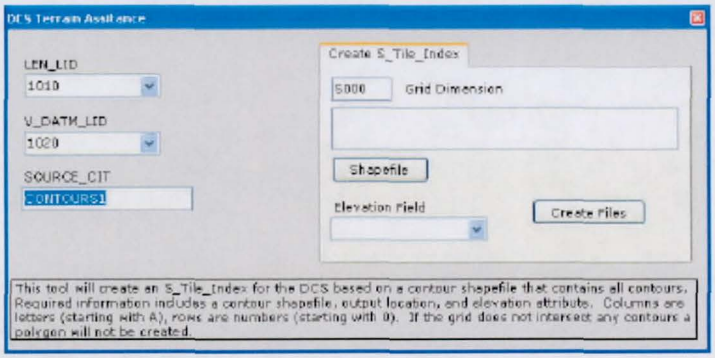


- Confirm the panel number.
- Select OK or cancel to exit.
- A note is added below the panel window.

Create a terrain tile index

- Select Tile Index from the Terrain DCS dropdown menu. 

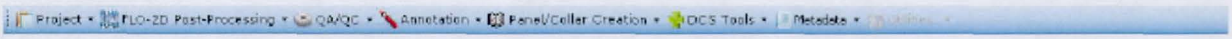
- This opens the Create Tile Index Dialog.



- This tool takes a contour file and breaks it up into a tiling scheme. It requires one input shapefile.
- The Grid Dimensions are in the unit of the shapefile.
- Select the Elevation field, and select the reset of the information on the left.
- Click Create Files. Errors are shown in a dialog box at the end of creation. Sometimes networks can cause problems with files.
- The files are outputted in the DCS Terrain directory under the project folder.

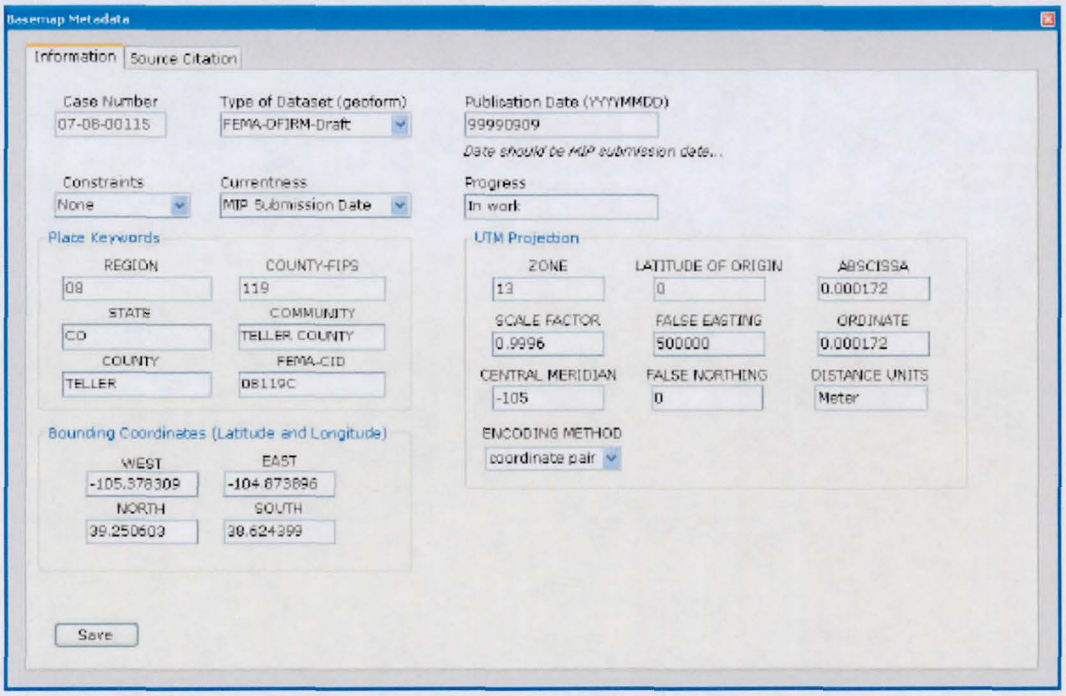
General Metadata Information

Metadata is stored in the project XML file. The metadata info tool is how you add and edit this information.



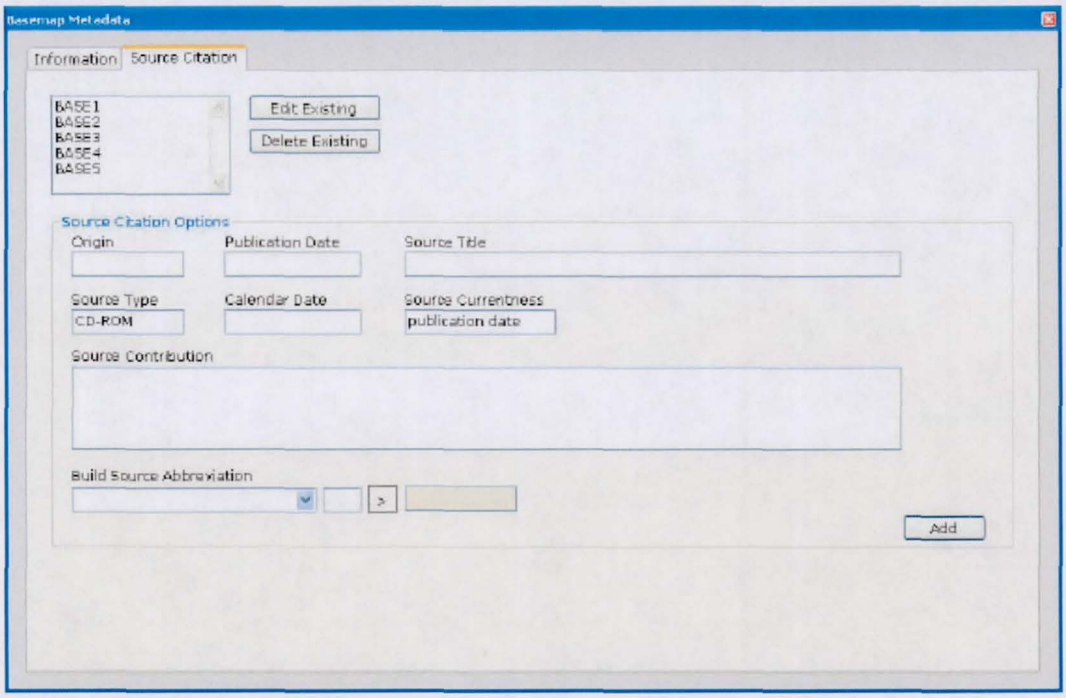
- Select Metadata Information from the Metadata dropdown menu.

- *This opens the Metadata Information dialog*



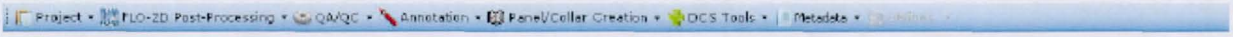
- This tab contains all the reuseable general information. Some fields will already be populated.
- Click save when changes have been made.

- This opens the Metadata Information dialog

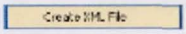


- This tab contains all source citation entries.
- *Select a source in the textbox and click Edit Existing* to change a source. *Click Add* to save it. Anytime there is the same Source Abbreviation, then the new informat information.
- Just enter information to add a new source.
- When you create an XML file, only the acceptable source citations are added.

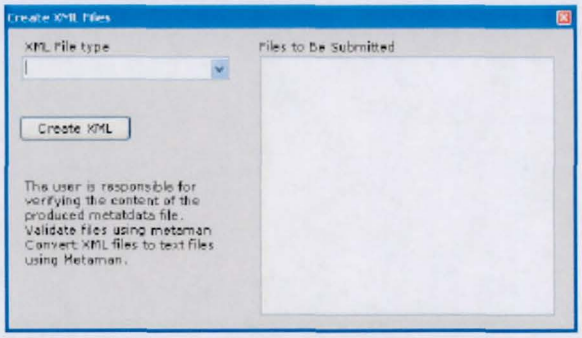
Create a metadata XML file



- Select Create XML from the metadata dropdown menu.

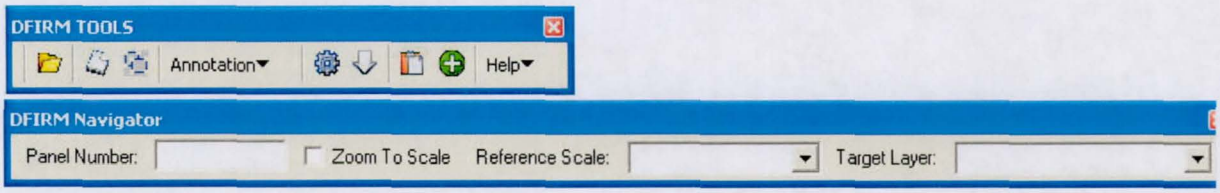


- This opens the XML File dialog.



- Select which metadata submission you are creating.
- Depending on which selection you will be asked to check off which files are submitted.
- A new XML file is created in the MSC Deliverables folder under the project folder.
- Be sure to check the file in Metaman, and also you can use metaman to convert to a txt file for the submission.

The DFIRM Toolbar

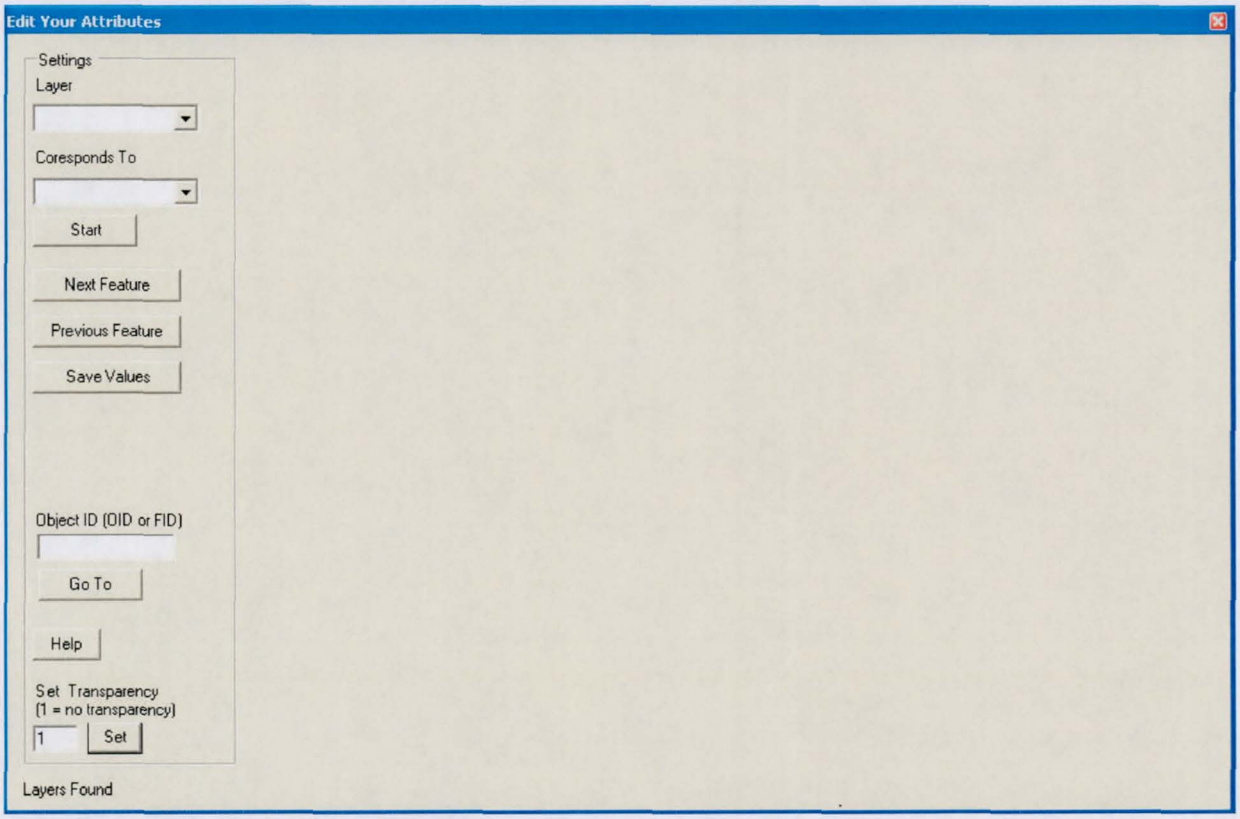


- There are two toolbars installed with ArcMap: DFIRM Tools and DFIRM Navigator.
- Opens a project folder. This will be required for the Annotation tools, Grid/Graticule, Township Range Note and Add 1000x1000 grid.
- The Attribute Editing tool is discussed here.
- This will populate the NW_LAT, NW_LONG, SE_LAT, SE_LONG fields. It requires these fields to exist in the database, but does not require the file to be called S_FIRM_PAN. The results should always be verified.
- Annotation is discussed here.
- Graticule/Grid and Township Range Note is discussed here and here.
- This tool will convert degrees minutes seconds to decimal degrees.
- This tool creates a shapefile of points, either a 1000X1000 (for UTM) or a 5000X5000 (State Plane) grid. It bases the information on the S_Quad_Index File, and uses that as the output folder.

The DFIRM Tools toolbar has a dialog to assist with attribute editing.

- Select the Edit Attribute button.

- This opens an input box.



- Select the layer you wish to edit from the drop down.
- Select what DFIRM DB the layer corresponds to. The name does not need to be the same.
- Click start. The DFIRM structure will appear on the right, and the first feature will be zoomed to. Missing fields will be listed at the bottom. The structure of the field does not need to be the same, but the name does.
- You can navigate one feature at a time using Next and Previous Feature.
- Edit values on the right side, then click save values to save them. There is no undo, so it isrecommended you make changes during an editing session, so if there is a change you do not have to save the edits. Otherwise, the changes are permanent.
- You can zoom to the Object ID of any feature.
- You can set the transparency of the dialog using decimals. 1 is equal to no transparency, and .5 would be equal to half transparancy. This makes it possible to view the features behind the dialog.

Converting Shapefiles to Coverages in ArcInfo 9.2:

1. When you export a shapefile to a coverage, the projection information is converted with it, but the TIC and Boundary information is not. A way around this is to conv ArcCatalog, and then use ArcInfo Workstation to do another conversion using the shapearc command line. The problem with using the commandline is that the projec coppied over, so just copy the dblbnd.adf and dbltic.adf files into the ArcCatalog converted files (in the folder).
2. Add the labels using ArcCatalog's Coverage toolbox to the polygons.
3. Verify the Correct Field Widths and Types (see below list)
4. Make sure that the feature count in the coverages are the same as the shapefiles
5. Covert to E00 format.

S_BFE

- FIELD: ELEV
- CHANGE TO NUMERIC 8,13,2

S_FIRM_PAN

- FIELD: EFF_DATE
- DATE 8,10

S_FLD_HAZ_AR

- FIELD: STATIC_BFE
- NUMERIC 8,13,2
- FIELD: DEPTH
- NUMERIC 8,13,2
- FIELD: VELOCITY
- NUMERIC 8,13,2
- FIELD: DEP_REVERT
- NUMERIC 8,13,2

S_LABEL_PT

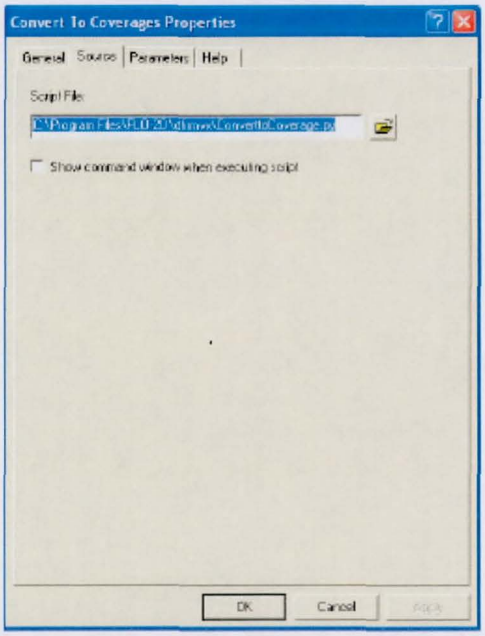
- FIELD: DEGREES
- BINARY 4, 10

S_XS

- FIELD: WSEL_REG
- NUMERIC 8,13,2

Geoprocessing Scripts for version 9.2:

Within the DFIRM installation directory under SupportFiles there is a toolbox and scripts that can be added to ArcMap and they will assist in the creation of the coverages. I this program, these require ArcInfo Workstation to be installed. You add the toolbox (FLO-2D DFIRM.tbx) by right clicking on the main toolbox and then add toolbox. Na directory and select just the toolbox. Once it has been added right-click on each script, select properties, and verify that it points to the correct location of that *.py file.



Converting Geodatabase Feature Classes to Shapefiles:

When you export a Geodatabase Feature Class to a Shapefile, the field width and field type information does not match the specification. Use the append tool in the toolbox t

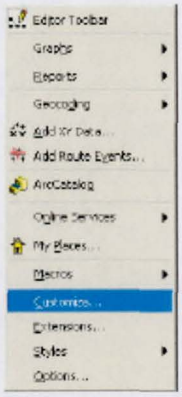
an empty shapefile. As long as the field name matches, each field will be populated correctly. The DFIRM tools can create empty shapefiles that match FEMA specifications

Automatically Update The Notes to Users Section everytime a file is opened:

Why automatically update? This makes it easier to have a change carry over throughout the entire project rather than individually changing each map. You still need to open take place, but this should make it easier.

1. Start by opening the template for your project. Make any changes you need to the original Notes to Users section you need and then copy all the text to a textfile (using not

2. For ArcMap 9.2 users. Right-Click on the Notes to Users Paragraph and select properties. Under the Size and Position tab you have the option to name the element. This s you can rename it if need be.



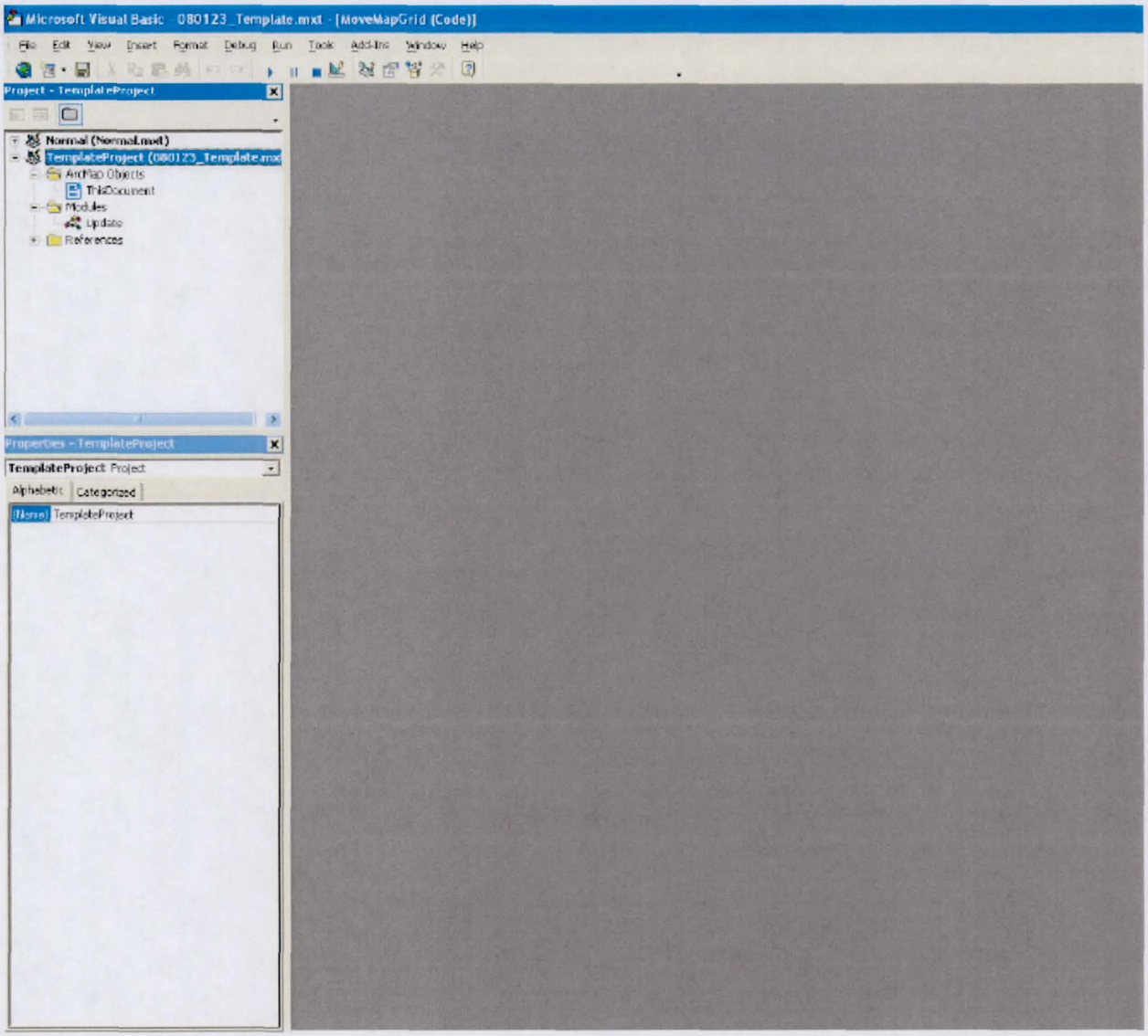
3. Under the Tools menu select Macros, then Visual Basic Editor

Open the TemplateProject, then ArcMap Objects. Double click on ThisDocument. Unless you've already added some code, it should be completely blank. Copy the followir code area. Change the path/filename to where you saved your text file

```
Private Function MxDocument_OpenDocument() As Boolean

Call UpdateNotesToUsers("X:\Projects\COCWCB06_Montezuma DFIRM\080123_DFIRM_Proj\Panels\notestousers.txt", "NotesToUsersBody")

End Function
```



Symbolizing Your Map:

For most objects that will be shown on the panel, using the domain values stored in the fields will suffice. For others, you can use one symbol for the whole layer. Others, y separate symbol field to make things easier. Included with your installation of the FLO-2D DFIRM Tools is a *.style called DFIRMv2.style. These are styles that the tools u the panels as well as the annotation. You can manipulate these and add your own to the style library. The following are some tips to help you symbolize your map.

- The S_BFE layer is straightforward, each bfe has the same symbol (called S_BFE in the library)
- S_FLD_Haz_Ar can be a bit more complicated. If you do not have floodways, then access your layers properties, and click the symbology tab. Under the Categories (Symbols in a Style. For the Value Field select FLD_ZONE and the style DFIRMv2.style. This should match all the zones to their appropriate field.
  If you do have floodways, use the Categories option again, but select Unique Values Many Fields. The first field should be FLD_ZONE and the second field should b the values and then symbolize accordingly.
- S_FLD_Haz_Ln, again, is complicated. With the need for gutterlines and the normal line symbols there are different options. You could create a graphic line at the lo but then these cannot be updated very easily. A better option is to add a field for symbols, and add a "GUTTERLINE" value for every gutterline (ie. zone breaks, and Then using the Unique Values Many Fields, select the LN_TYP and SYMBOL fields. Now you can maintain the integrity of your spatial database, but symbolize you
- S_Gen_Struct is actually tied to two other spatial databases: S_Wtr_Ln, and S_Trnsport_Ln. Since you do not want to see the water line (in the case of a culvert) or th a bridge), then these lines need to be hidden. An easy way to do this is to create a symbol field. At the end points of the gen_struct, break either the wtr_ln or trnsport object using the symbol field. For example, a bridge as "BRIDGE", a culvert as "CULVERT", etc. Then under the S_Trnsport_Ln and S_Wtr_LN layer properties, us exclude any feature that has something in the SYMBOL field. Now you can symbolize the S_Gen_Struct using the Match Symbols in a Style option.
- S_Label_Ld is dependant on how you decide to populate your shapefile or feature class.
- S_Perm_Bmk uses one symbol for all features.
- S_PLSS_Ln is straight forward; use Match Symbols in a Style for the LN_TYP field.
- S_POL_Ln is straight forward; use Match Symbols in a Style for the LN_TYP field.
- S_Trnsport_Ln is straight forward; use Match Symbols in a Style for the TRANS_TYP field. You should also review the S_Gen_Struct section above.
- S_Wtr_Ln is straight forward; use Match Symbols in a Style for the WATER_TYP field. You should also review the S_Gen_Struct section above.
- S_XS uses one symbol for all features.
- Note: You can add any of your completed files to the template before generating your panels. These are stored in the template, and hence will be in your mxd files wh is a quick of way have having them in your mxd file. Once an mxd file, any changes to the template are not applied to that mxd file.

Editing the support files:

The FLO-2D DFIRM tools has attempted to make it easy to edit the support files to customize the user's working environment. For example: Creating Shapefiles, Attribute Table Checks, Creating Tables, and Attribute Editing all refrence two files for their proecesses. If there are files that the user needs to generate (such as the coastal DFIRM files) that are not already available in the display, these can be added to the dfirm.xml file and the user then has access to them. They should also be added to the project xml file as well.

- Locate the dfirm.xml file in the installation folder for the DFIRM Tools. Example: C:\Program Files\FLO-2D\DFIRM\SupportFiles\dfirm.xml

Files\FEMA\DFIRM\Support Files\dfirm.xml.

- Right-click on the dfirm.xml file and select open with WordPad or NotePad.


```
<?xml version="1.0"?>
<DFIRM>
	<table name="S_BFE" type="3">
		<field>
			<aliasname>BFE_LN_ID</aliasname>
			<type>4</type>
			<length exists="T">11</length>
			<precision exists="F"></precision>
			<scale exists="F"></scale>
			<lookup exists="F"></lookup>
			<required>R</required>
			<pk>T</pk>
		</field>
		<field>
			<aliasname>ELEV</aliasname>
			<type>3</type>
			<length exists="F"></length>
			<precision exists="T">13</precision>
			<scale exists="T">2</scale>
			<lookup exists="F"></lookup>
			<required>R</required>
		</field>
		<field>
			<aliasname>LEN_UNIT</aliasname>
			<type>4</type>
			<length exists="T">20</length>
			<precision exists="F"></precision>
			<scale exists="F"></scale>
			<lookup exists="T">D_Length_Units</lookup>
			<required>R</required>
		</field>
		<field>
			<aliasname>V_DATUM</aliasname>
			<type>4</type>
			<length exists="T">6</length>
			<precision exists="F"></precision>
			<scale exists="F"></scale>
			<lookup exists="T">D_V_Datum</lookup>
			<required>R</required>
		</field>
		<field>
			<aliasname>SOURCE_CIT</aliasname>
			<type>4</type>
			<length exists="T">11</length>
			<precision exists="F"></precision>
			<scale exists="F"></scale>
			<lookup exists="F"></lookup>
			<required>R</required>
		</field>
	</table>
```


- The above image is selected from the installed dfirm.xml file and is for the S_BFE table. Each table ha a start <table> and an end </table>. The start has two attributes "name" and "type". Name obviously is the name of the table, and type is what type of shapefile it refers to (point, polyline, or polygon). You can see that the type in this example is actually a number. Polyline = 3, Polygon = 4, and Point = 1.
- The next "sub" element is the "field" element, and each table will have at least one field element. These fields correspond to the final DFIRM database structure provided ing FEMA's appendix L.
- Each field has at least 7 subelements: aliasname, type, length, precision, scale, lookup, required, and sometimes pk.

- "aliasname" is the name of the field. "type" is the type of field. The most common ones will be double string (text), or date. Again you see in the example it is a number: Double = 3, String = 4, Date = 5
- Depending on what type of field you are creating, the next three options will have different settings. A double type uses both precision and scale, so the exists attribute is set to "T", and the length exists is set to "F". Precision is the width of the field, and scale is how many decimal places. For a string type length is set to "T" and precision and scale are set to "F". The same for Date.
- Certain fields have domain tables associated with them. The lookup element is set to "T" and then the name of the lookup is entered. The domains are stored in the SupportFiles folder in a file called domain.xml. Here is an example:

```
<?xml version="1.0"?>
<domain>
        <table name="D_Area_Units">
                <value>ACRES</value>
                <value>HECTARES</value>
                <value>SQUARE FEET</value>
                <value>SQUARE METERS</value>
                <value>SQUARE YARDS</value>
        </table>
        <table name="D_Chan_Rep">
                <value>SINGLE</value>
                <value>DOUBLE</value>
        </table>
        <table name="D_Floodway">
                <value>FLOODWAY</value>
                <value>COLORADO RIVER</value>
                <value>FLOODWAY CONTAINED IN CHANNEL</value>
                <value>FLOWAGE EASEMENT BOUNDARY</value>
                <value>STATE ENCROACHMENT</value>
                <value>AREA OF SPECIAL CONSIDERATION</value>
                <value></value>
        </table>
        <table name="D_Label_Typ">
                <value>DOQ-TRANSPORTATION</value>
                <value>DOQ-WATER</value>
                <value>S_TRNSPORT_LN</value>
                <value>S_WTR_AR</value>
                <value>S_WTR_LN</value>
        </table>
        <table name="D_Length_Units">
                <value>CENTIMETERS</value>
                <value>FEET</value>
                <value>INCHES</value>
                <value>KILOMETERS</value>
                <value>METERS</value>
                <value>MILES</value>
                <value>KILOMETERS</value>
        </table>
```

- As you can see it is similar to the dfirm.xml file. The table element with the table name is there. Then each value that is in the domain follows the table element. The table name is what goes between the lookup opening and closing brackets: <lookup exists="T">D_V_Datum</lookup>
- The only entries that should be used for required are "R" and "A" which is in Appendix L.
- pk is added for primary key fields.
- Save your changes. Errors will occur if you have made incorrect edits. The easiest will be to copy exisiting settings and change them for your requirements.

Troubleshooting

We appologize for any problems that might arise during the use of the DFIRM tools. If you have any problems, questions or comments, we will try to assist you. Please contact us at dfirmsupport@acewater.com. There are no expressed or implied warranties associated with this tool. The tools will be refined and enhanced with applications to unique and complex flood mapping.

The most common problem to arise will be with the install of the toolbars. The way the toolbars work, is the register with Global .NET Assemblies, and this is registered with ArcMap. That is a very short explanation for something more complex. Normally this is taken care of by the installer and uninstaller. Depending on how you installed your version of ArcGIS 9.2 you might not have the correct settings.

- Check to make sure you have the latest service pack for ArcGIS 9.2. At the time of release this was service pack 3. Certain of the .NET assemblies were updated in the latest service pack, and the program is written for this service pack.
- If the service pack is fully up-to-date, you can manually register the toolbar. First identify where the regasm.exe file lives in the .NET 2.0 folder. The default installation is here: C:\WINDOWS\Microsoft.NET\Framework\v2.0.50727. Next go to the start menu and select Run. In the command prompt enter the full path to the regasm including the regasm.exe file followed by a space. Add the default FLO-2D DFIRM tools path followed by DFIRM_Toolbar.dll space /tlb space /codebase. It will look something like this: C:\WINDOWS\Microsoft.NET\Framework\v2.0.50727\regasm.exe C:\Program Files\FLO-2D\DFIRM\DFIRM_Toolbar.dll /tlb /codebase then click ok. To manually unregister the dll during uninstall: C:\WINDOWS\Microsoft.NET\Framework\v2.0.50727\regasm.exe C:\Program Files\FLO-2D\DFIRM\DFIRM_Toolbar.dll /u
- If that did not work, uninstall the FLO-2D DFIRM tools (you do not need to uninstall the actual FLO-2D program). Obtain a copy of your ArcGIS installation CD and rerun the installation. You do not need to fully reinstall, but you do need to add the .NET assemblies. Once this is complete, rerun the service pack, and then install the FLO-2D DFIRM tools.

This article from Microsoft shows how to force the removal of the older package using the Windows Installer CleanUp Utility.
http://support.microsoft.com/kb/290301