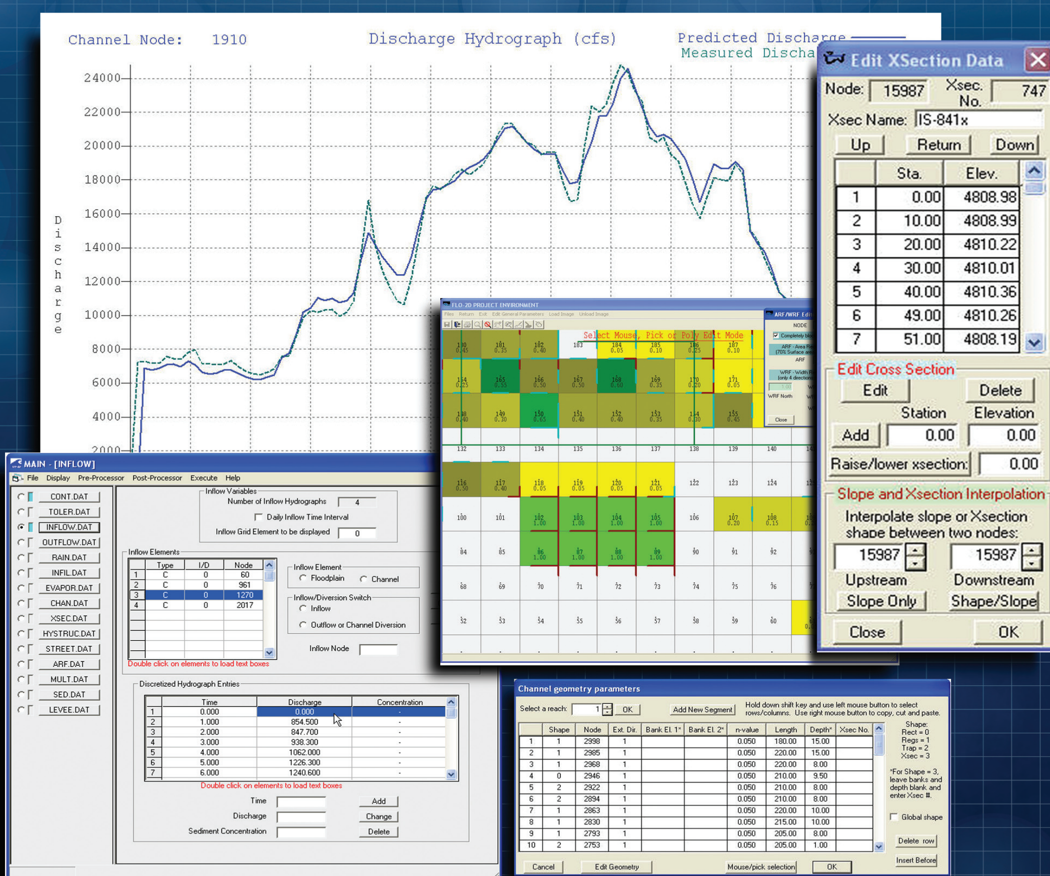
# FLO-2D

# Data Input Manual

August 2019 - Build 19



# PREFACE

## FLO-2D INPUT DATA OVERVIEW

This manual describes the FLO-2D Pro Model data input parameters and their format. The FLO-2D data base 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, levee or dam breach 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, eight data files must be created. These files can be automatically generated using the grid developer system (GDS) program. The six required files for basic overland flow simulation are:

- TOPO.DAT
- MANNINGS_N.DAT
- FPLAIN.DAT
- CADPTS.DAT
- CONT.DAT
- TOLER.DAT
- INFLOW.DAT
- OUTFLOW.DAT

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). These files can be used in lieu of the FPLAIN.DAT and CADPTS.DAT files. 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 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 NotePad®, 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 two ways to run a FLO-2D simulation once the data files are constructed. 1) The Pro model can be initiated from the GDS/QGIS; or 2) a FLO-2D flood simulation can be started by copying the FLOPRO.EXE file and its respective dlls into a project folder and double-clicking on the executable. 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, CRAYFISH, 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

## List of Tables

# 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, preprocessor programs, data files, output files, post-processor programs and troubleshooting. Chapter 4 can be useful 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 use with Windows 7 or 8 computers with 64-Bit operating systems, multiple processors 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 data that can be interpolated 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

The FLO-2D Pro model has been compiled for 64 bit multi-core processor computers. It cannot be run on a 32-bit computer. Recommended minimum computer requirements are at least 4 GB RAM. 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. The default directory is C:\PROGRAM FILES(x86)\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\FLO-2D PRO Documentation. 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 appears to keep shared DLL/OCX files, 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 the Program Files (x86) folder. If FLO-2D folder contains project files, move them or back them up before un-installing or re-installing the software.

## 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 them 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 Pro Documentation\flo_help). It has step-by-step instruction of how create and edit detailed flood components.

C:\users\public\public documents\flo-2d pro documentation\flo_help\workshop lessons

### Seeking Assistance – Technical Support

Hint:
**When sending data files for tech support, zip together only *.DAT files and send them to contact@flo-2.d.com**

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 components that are not contributing to 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.

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, point 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 Grid, ASCII xyz data sets and ArcGIS elevation shapefiles.

**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 from a watershed model can be the desired product or the watershed model 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 and rainfall 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 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.

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:

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

**Table 1.1. Grid System Size**

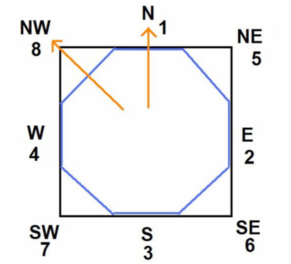
| Number of Grid Elements | Model Simulation Speed |
|---|---|
| < 50,000 | Fast (minutes) |
| 50,000 – 250,000 | Moderate (<12 hours) |
| 250,000 – 1,000,000 | Slow (> 12 hours) |
| > 1,000,000 | Very Slow (> 1 day) |

The closer $Q_{peak}/A_{surf}$ is to 3.0 cfs/ft$^2$ (0.1 cms/m$^2$), the faster the model will run. If the $Q_{peak}/A_{surf}$ is much greater than 10.0 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 built.

A suggested order of component construction is as follows:




- Rainfall/Infiltration
- Channels
- Levees
- Streets
- Buildings
- Hydraulic Structures (culverts, weirs and bridges)
- Storm Drains
- 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.

## Saving data

When creating or editing the data files, it is suggested that the data files saved frequently and that one folder for testing a project and another one for editing a project. It is suggested that the data files be saved after finishing each component.

## Develop the Project Files

*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 graphically create and edit the grid system Follow the GDS "Getting Started" lesson.

Hint:

**Basic Data Files:**

- FPLAIN.DAT
- CADPTS.DAT
- CONT.DAT
- TOLER.DAT
- TOPO.DAT
- MANNINGS_N.DAT

Optional

- INFLOW.DAT
- OUTFLOW.DAT

*Run the FLO-2D model*

The required data files for a basic overland flood model are:

- FPLAIN.DAT
- CADPTS.DAT
- CONT.DAT
- TOLER.DAT
- INFLOW.DAT
- OUTFLOW.DAT
- TOPO.DAT
- MANNINGS_N.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.

**Hint:**
**Test initial model runs with a simple steady state inflow hydrograph.**

## Some General Guidelines

*Data Input*

When the data format appears confusing, review the data files provided in the Example Projects subdirectory of the FLO-2D folder using an ASCII editor such as NotePad++©.

*File Management*

The output files are always generated with the same name and will be overwritten in subsequent model runs. To save any output files that could 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 view the floodwave progression during the simulation, run the simulation 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.

*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 Model Component Considerations

### Channel Modeling

The 1-D channel component can simulate flow in channels defined by various geometries.  The flow shares between the channel banks and the floodplain.  Channels are defined in FLO-2D whenever 1-D flow is more accurate than overland flow. They can reduce flooding and help the water move downstream mare quickly than flow on the floodplain.  An extensive Channel Guidelines document is available in the Manuals Folder.  C:\users\Public\Public Documents\FLO-2D Pro Documentation\flo_help\Manuals.

### 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.

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.**

### 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 floodwave 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**

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, a HEC-RAS cross section data file is automatically converted to a FLO-2D XSEC.DAT cross section file.

*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 a HEC-RAS channel and cross section data is overlaid onto a FLO-2D grid system. The steps required to create a FLO-2D natural channel data base from HEC-RAS data are followed.

*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*

The procedure for setting up a dam breach erosion model is presented in this lesson.

*Lesson 13 – Multi-Domain Interface (Outflow to Inflow)*

This lesson prepares the outflow discharge hydrograph from one grid system as the inflow hydrograph to another grid system and describes how to set up the outflow nodes that will automatically capture the upstream grid system hydrograph. 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.

*Lesson 15 – Prescribed Levee Breach*

This lesson outlines the process of setting up and testing simple levee prescribed breach criteria and performing test runs.

*Lesson 16 – Simple Storm Drain*

This lesson outlines the process of setting up and testing simple storm drain using GDS and EPA SWMM 5.0.

*QGIS Lesson 1 – Getting Started Urban*

This lesson outlines the process of setting up a urban FLO-2D project. It includes building a GeoPackage, setting the project domain, creating a grid, interpolating elevation data, setting n-value data and creating buildings in QGIS using the FLO-2D Plugin.

*QGIS Lesson 2 – Channels, Culverts and Walls Urban*

This lesson outlines the process of setting up channels, hydraulic structures and defining walls in QGIS using the FLO-2D Plugin.

*QGIS Lesson 3 – Storm Drain Urban*

This lesson outlines the process of converting a storm drain network from shapefile into a FLO-2D Storm Drain system in QGIS using the FLO-2D Plugin..



## 2.2 FLO-2D Example Flood Simulations

A number of example projects are provided in the FLO-2D Documentation/Example Projects subdirectory. To run these projects, either load them in the GDS or run them the project folder (first make sure that the FLOPRO. EXE file is in the project subdirectory and double click on the FLOPro.exe name). Most of the example simulations are setup for the graphics mode and will take only a few minutes to run.

*Working with Geo-referenced Images – Goat Camp Creek, Gila County, Arizona (Goat subdirectory)*

This 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.

*Large River Flooding – Rio Grande, New Mexico (Rio Grande subdirectory)*

Over 173 miles of the Middle Rio Grande is simulated using surveyed channel cross section data. 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 and river-floodplain discharge exchange.

*Rainfall/Runoff - Blue Diamond Watershed, Nevada (Diamond subdirectory)*

Hint:
**Review the Example Project data files that are similar in concept your project.**

Rainfall, infiltration and runoff are simulated for a 34 square mile desert Rainfall, infiltration and runoff are simulated for a 34 square mile desert watershed. Steep, unvegetated ridges, bedrock and numerous small watershed ravines characterize the project area. The drainage pattern can be observed as the rainfall/runoff progresses 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.

*Urban Watershed Flooding – Waikiki Beach, Oahu, Hawaii (Alawai subdirectory)*

This urban flooding example includes 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.

*Alluvial Fan Channel Overbank Flooding - Monroe Creek Alluvial Fan, Utah (Monroe folder)*

This example flood simulation depicts channel-floodplain discharge exchange. The channel is represented by variable channel geometries including rectangular, trapezoidal and natural-shaped. The overbank flooding can be observed returning to the channel. This simulation is a good example for reviewing the variable channel geometry data in the CHAN.DAT file.

*Urban Fan and 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 mudflow enters the grid system at 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.

*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.

*Urban Shallow Flooding - Urban Project Example*

Small isolated portion of a large urban study. This project has examples of trapezoidal channels, culvert, walls, buildings, urban n-values and a simple storm drain system.

*Storm Drain Example - Storm Drain*

Small isolated portion of a large urban study. This project has a complete working storm drain system.

*Sediment Transport - Sediment Transport Channel Example*

Fully functional example of sediment transport routing in a 1-D channel.

## 2.3 FLO-2D PowerPoint Presentations

These presentations discuss most of the FLO-2D components. The files can be found in FLO-2D Documentation\flo_help\PowerPoint Presentations.

## 2.4 Other Help Documents

Several documents in the FLO-2D Handout folder provide advanced model guidance and discussion of specific components and model techniques. They can be found in FLO-2D Documentation\flo_help\Handouts.

## 2.5 Metric Option



Hint:

**Copy the following Processor Programs into a project folder. Double click the executable name to run the program:**

- **FLO.EXE**
- **PROFILES.EXE**
- **MAXPLOT.EXE**
- **HYDROG.EXE**

The user can choose either the English or Metric system of units (for the Metric

| 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 |

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:

Manning's n-Value is considered an Imperial number and it is not necessary to convert it for Metric or English units. The conversion is part of the calculation.

# CHAPTER 3

## Pre-Processor Programs

### 3.1 Introduction

There are three pre-processor programs to help to create or edit the FLO-2D data files: QGIS Plugin, 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 QGIS FLO-2D Plugin

The FLO-2D Plugin for Quantum Geographical Interface System (QGIS) is a program developed to generate FLO-2D data files using QGIS. This program has separate documentation available in the FLO-2D Documentation\flo_help subdirectory.

### 3.3 GDS

The grid developer system (GDS) is a GIS program used to create and edit the FLO-2D grid system and its attributes. 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.

### 3.4 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 PROFILES.EXE to the project folder and double click it.

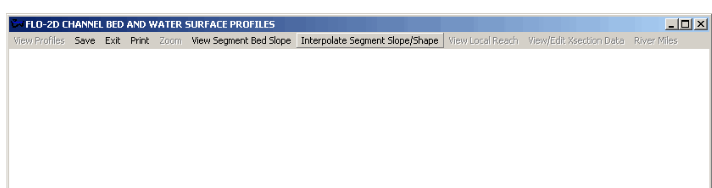
Before using the PROFILES program, the basic FLO-2D files plus the CHAN.DAT file have to be created. The XSEC.DAT will also have to be created if surveyed cross section data will be used. The general procedure for using the PROFILES program is as follows:

> Hint:
> **To quickly access the PROFILES program, copy the executable PROFILES.EXE into the project folder 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. Save data in PROFILES using the Save menu. This option is activated after an edit has been made. The save option allows for two datasets. The data can be overwritten or saved as new. The data is not written to file until PROFILES is closed.

> 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 0.**

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'.



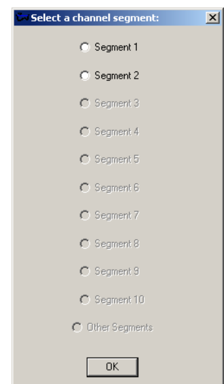
## Interpolating a New Channel with Surveyed Cross Sections

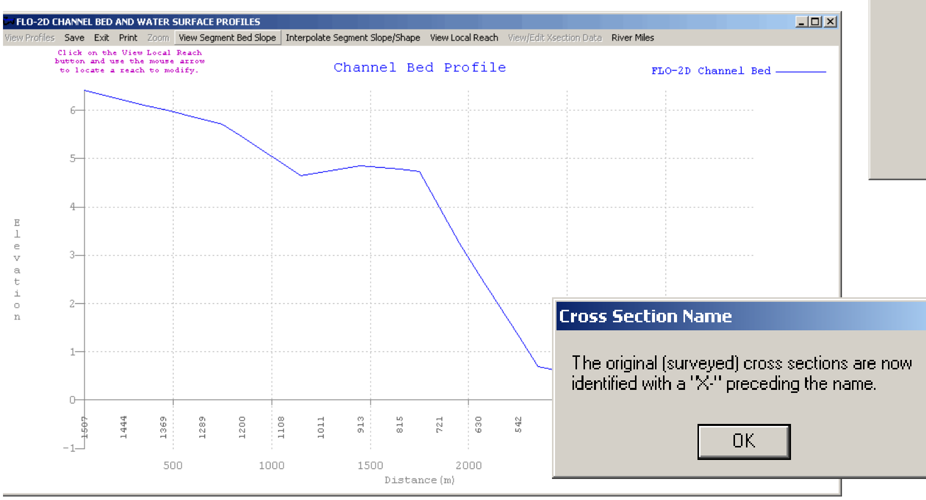
To interpolate the cross sections and slope and assign a cross section to every channel element in PROFILES, use the 'Interpolate Segment Slope/Shape' menu option as follows:

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 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.

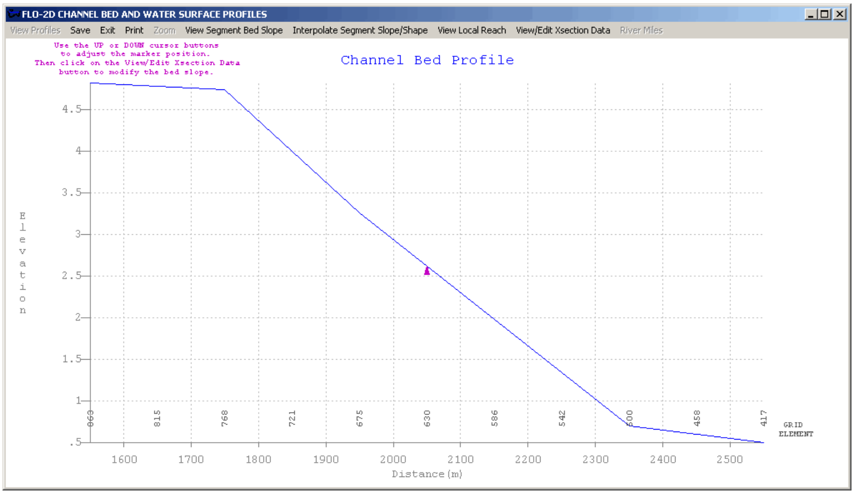
2. PROFILES will automatically locate the surveyed cross section data and interpolate the cross section geometry and elevation (thalweg slope) for all those channel elements between the surveyed cross sections within the segment. The following dialog box will appear indicating that the original cross 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.



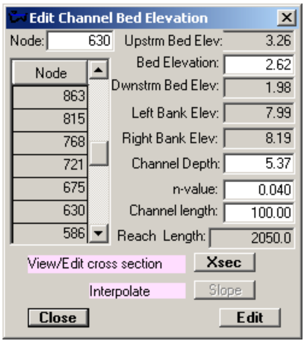

4. Click on the 'View Local Reach' button on the menu bar. Click anywhere along the bedslope profile to zoom in on a local reach of 10 channel elements.



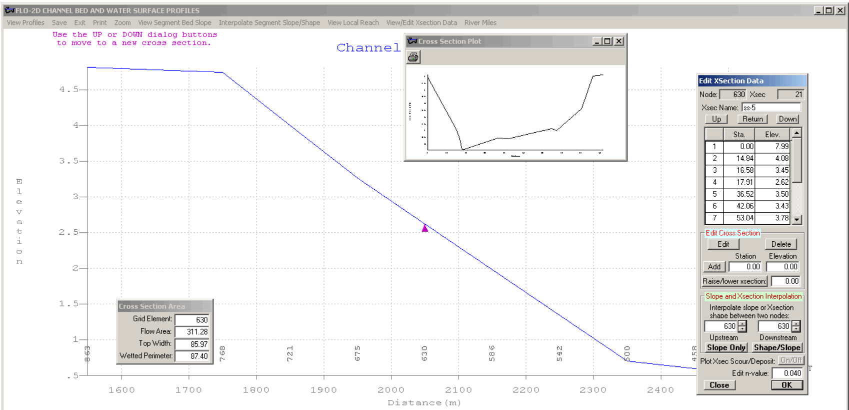

**Note:**

**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.




7. View additional 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. 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.

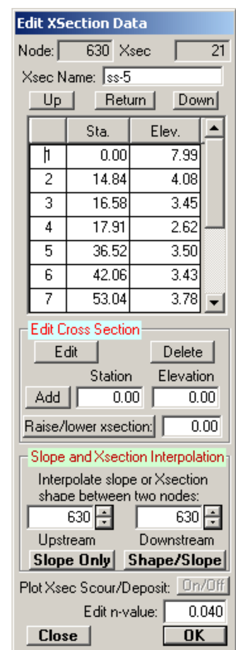
8. Interpolate bedslope and or channel geometry. Identify the Upstream and Downstream channel elements in the group boxes labeled 'Slope and Xsection Interpolation'. Use 'Up' and 'Down' buttons to locate one of the surveyed cross sections and then type in the other either upstream or downstream channel 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. The cross section geometry is linearly interpolated according to top width and distance and is adjusted for the weighted flow area. One cross section is overlaid on the other cross section, stretched or contracted and the elevations averaged.

9. Save the results frequently by clicking on 'Save' on the menu bar. The saved data will not be written to file until the program is closed.

10. NOTE: Perform the initial channel interpolation of the cross sections automatically in the GDS or QGIS.

Hint:

When interpolating between two cross sections, they must be entered in order from upstream to downstream..



# INPUT DATA FILE DESCRIPTION

## 4.1 GENERAL

The FLO-2D data file variables and format are described in this chapter. These files are accessed directly by the model. 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 variable descriptions for clarification. The GDS/QGIS or any ASCII text editor can be used to create or edit the data files.

All of the data entries, integers (i) or real numbers (r) are in free format space delimited. The ID characters (letters) are case sensitive. The variables are listed line by line and 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 (array indices) represent element number and hydrograph pair;
**H(I,J,1)** = time (hrs) start of the discretized interval of inflow hydrograph;
**H(I,J,2)** = discharge (cfs);
**H(I,J,3)** = mudflow sediment concentration by volume 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 Line 3 data 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.

## 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 and referring to these numbers may help to locate input data errors.

## 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 typically are necessary for 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 or QGIS and does not require editing. The TOPO.DAT and MANNINGS_N.DAT files are generated by the GDS or QGIS and the FLO-2D Pro model at runtime. The TOPO.DAT and MANNINGS_N.DAT files replace the FPLAIN.DAT and CADPTS.DAT file (are obsolete and not necessary but can still be used).

| File No. | File Name | File No. | File Name |
|---|---|---|---|
| 37 | ARF.DAT | 311 | MANNINGS_N.DAT |
| 300 | BATCHCYCLE.DAT | 38 | MULT.DAT |
| 250 | BREACH.DAT | 50 | OUTFLOW.DAT |
| 287 | BUILDING_COLLAPSE.DAT | 98 | OUTRC.DAT |
| 10 | CADPTS.DAT | 1651 | QGISDEBUG.DAT |
| 162 - 170 | CADPTS_DS1-9.DAT | 32 | RAIN.DAT |
| 36 | CHAN.DAT | 89 | RAINCELL.DAT |
| 119 | CHANBANK.DAT | 1568 | SDCLOGGING.DAT |
| 30 | CONT.DAT | 39 | SED.DAT |
| 990 | DEBUG.DAT | 1608 | SHALLOWN_SPATIAL.DAT |
| 95 | EVAPOR.DAT | 52 | STREET.DAT |
| 125 | FPFROUDE.DAT | 1557 | SWMMFLO.DAT |
| 31 | FPLAIN.DAT | 1559 | SWMMFLORT.DAT |
| 31 | FPLAIN.DAT | 1562 | SWMMOUTF.DAT |
| 120 | FPXSEC.DAT | 211 | TIMDEPCELL.DAT |
| 200 | GDSRUN.DAT | 9 | TOLER.DAT |
| 68 | HYSTRUC.DAT | 1600 | TOLSPATIAL.DAT |
| 33 | INFIL.DAT | 97 | TOPO.DAT |
| 34 | INFLOW.DAT | 180 | WSTIME.DAT |
| 171 - 179 | Inflow1-9_DS.DAT | 85 | XSEC.DAT |
| 57 | LEVEE.DAT | | |
| | | | |
| | | | |
| 1575 | SWMM.INP | | NEIGHBORS.DAT |
| 1558 | SWMM.RAIN | | |
| 100 | used | 114 | used |
| 1993 | used | | |

Table 4.1. List of *.DAT Files and Unit Numbers

# 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 3.0 | Line 6: | **NOPRTFP DEPRESSDEPTH** |
| 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 8: If ITIMTEP = 5 TIMEDEPCELL.DAT is required
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 3.0
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** | **-1 to 1**<br>**-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 or <-1.0 to globally increase or decrease the n-value multiplicatively (n-value * AMANN). |
| DEPRESSDEPTH | **r** | **0.0 - 10.0** | DEPRESSDEPTH identifies depressed grid elements that are lower than all contiguous nodes. A value of DEPRESS-DEPTH = 3.0 ft is suggested. Depressed elements may be real, but in most cases isolated depressed elements are a result of poor topographic data (see comments 10 thru 12). |
| DEPTHDUR | **r** | **0.01 - 100**<br>**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**<br>**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** | Limiting 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). The increased n-values are reported in the ROUGH.OUT and FPLAIN.RGH files (see comments 3 and 4). |
| GRAPTIM | **r** | **0.01 - 10.** | Time interval in hours that the graphics display is updated (e.g. set GRAPTIM = 0.02 for a frequent update). GRAPH-TIM 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. |

# Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IBACKUP | s | **0 = off**<br>**1 = on**<br>**2** | IBACKUP = 1 creates a backup file of all the data files with a *.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). |
| ICHANNEL | s | **0 = off**<br>**1 = on** | If ICHANNEL = 1, the channel component will be used and the CHAN.DAT must be created (comments 1 and 6). |
| IDEBRV | s | **0 = off**<br>**1 = on** | Set IDEBRV = 1 if a debris basin volume should be filled before routing the flow hydrograph. |
| IEVAP | s | **0 = off**<br>**1 = on** | Set IEVAP = 1 if simulating free water surface evaporation from overland or channel flow. |
| IFLOODWAY | s | **0 = off**<br>**1 = on** | If FLOODWAY =1, a floodway analysis will be performed in the subsequent FLO-2D simulation. An initial FLO-2D flood simulation must be completed prior to a floodway simulation (see comment 5). |
| IHYDRSTRUCT | s | **0 = off**<br>**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**<br>**1 = on** | Set IMULTC = 1 to simulate multiple channel (rill and gully flow) rather than overland sheet flow between multiple channel elements. The MULT.DAT file must be created. |
| IMODFLOW | s | **0 = off**<br>**1 = on** | Set IMODFLOW = 1 to simulate surface-groundwater exchange. This switch initiated the linked MODFLOW groundwater model a during the FLO-2D simulation. |
| INFIL | s | **0 = off**<br>**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**<br>**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**<br>**1 = on** | If ISED = 1, the sediment transport routine will be used. The SED.DAT file must be created. |

# Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| ITIMTEP | s | **0 = off<br>1, 2, 3, 4, and 5 = on** | 0 = No time series output is written at runtime.<br>1 = TIMEDEP.OUT is written at runtime.<br>2 = TIMEDEP.OUT and HDF5 files are written at runtime.<br>3 = NETCDF4 files are written at runtime.<br>4 = All time series output is written at runtime.<br>5 = Extract a time series for specific cells. Requires TIMDEPCELL.DAT |
| IWRFS | s | **0 = off<br>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<br>1 = on** | Set LEVEE = 1 to simulate levees. The LEVEE.DAT file must be created. |
| LGPLOT | s | **0 = text<br>1 = batch<br>2 = graphic** | LGPLOT = 0 will display screen text scrolling the simulation time, minimum timestep and volume conservation.<br>LGPLOT = 1 will display nothing. Use this switch position with batch runs.<br>LGPLOT = 2 displays the graphical floodwave progression over the grid system (flow depth) and inflow hydrograph. |
| METRIC | s | **0 = English<br>1 = Metric** | METRIC = 0 for English units and METRIC = 1 for the metric system of units. |
| MSTREET | s | **0 = off<br>1 = on** | MSTREET = 1 to initiate the street flow component. The STREET.DAT file must be created (see comment 2). |
| MUD | s | **0 = off<br>1 = on** | Set MUD = 0 for clear water and MUD = 1 for hyper-concentrated sediment flow. If MUD = 1 the sediment load (volume or concentration by volume) for either the floodplain hydrograph HP(I,J,3) or the channel hydrograph H(I,J,3) must be assigned to each inflow hydrograph pair (comments 1 and 3). The SED.DAT file must be created. |
| NOPRTC | s | **0, 1 or 2** | If NOPRTC = 0, the BASE.OUT channel data is reported.<br>If NOPRTC = 1, the BASE.OUT channel outflow data is not reported.<br>If NOPRTC = 2, the BASE.OUT file is not reported. |

# Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| NOPRTFP | s | **0, 1, 2 or 3** | If NOPRTFP = 0, the BASE.OUT floodplain flow data is reported.<br>If NOPRTFP = 1, the BASE.OUT floodplain outflow data is not reported.<br>If NOPRTFP = 2, BASE.OUT is not written. This reduces the time for writing model output.<br>If NOPRTFP = 3, only floodplain outflow data is reported to the BASE.OUT file. |
| SHALLOWN | r | **0 = off**<br>**0.1 - 0.99** | Flow roughness n-value for shallow overland flow (flow depth < 0.2 ft or 0.06 m) (see comment 9). |
| SIMUL | r | **0.01 - ∞** | Simulation time (hours). |
| SWMM | s | **0 = off**<br>**1 = on** | SWMM = 1 initiates the FLO-2D storm drain model. |
| TIMTEP | r | **0 - 100** | Output interval (hrs) that the flow depth, resolved velocity, x-velocity, y-velocity and water surface elevation datasets are reported to the TIMDEP.OUT file for a post-simulation flood animation. TIMTEP should be a multiple of TOUT. The switch ITIMTEP = 1 is required. |
| TOUT | r | **0.01 - 24.** | Output interval (hrs) that hydraulic data is reported to 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 of 0.10 indicates that 10% of each grid element surface is not available for flood storage. The XARF value is overridden by the ARF variable specified for the individual grid elements in the ARF.DAT file. Assign XARF = 0. to flood the entire surface area of the grid elements. |

# Variable Descriptions for the CONT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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 hyperconcentrated 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

These instructions will aid in assigning of the CONT.DAT file parameters:

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. Supercritical flow is uncommon on alluvial surfaces 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. Set the Limiting Froude number or the floodplain (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 a flood simulation, review ROUGH.OUT to determine where FROUDL was exceeded and the maximum n-values for that cell were computed. Consider revising the n-values in the MANNINGS_N.DAT file to match those in the ROUGH.OUT file. This will ensure that FROUDL is not exceeded. Rename the MANNINGS_N.RGH file to MANNINGS_N.DAT.

   d. Spatially variable limiting Froude numbers can also be assigned to individual grid elements in FPFROUDE.DAT.

   e. The shallow n-value is off when SHALLOWN = 0. or when AMANN = -99. The limiting Froude number is off if you set FROUDL = 0. for the floodplain. AMANN= -99 turns off the depth variable n-value, but not the limiting Froude number n-value adjustments.

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. An initial FLO-2D simulation is required to establish the maximum water surface elevations. See the Floodway discussion in the Reference Manual component section.

6. If channel flow is simulated (ICHANNEL = 1), then the NOPRTC variable must be set in CONT.DAT. In addition, channel outflow control can be assigned in OUTFLOW.DAT.

7. ITIMTEP will enable a simple animation (time and space output) of the overland flow to be displayed in Mapper, MAXPLOT or other map software. 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:

$$n_d = n_b * 1.5 * e^{-(0.4\ depth/dmax)}$$

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 elements (assumed to be 3 ft or 1 m). This equation is applied by the model as a default and the user can turn '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 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 about 1.31 times the assigned roughness for a flow depth of 3 ft. Assigning a ROUGHADJ value may reduce unexpected high Froude numbers.

The following rules apply:

If the

| | |
|---|---|
| 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) | $n = n_b * 1.5 * e^{-(0.4\ depth/dmax)}$ |
| 3 ft (1 m) < flow depth | n = n-value in MANNINGS_N.DAT |

10. The IBACKUP = 1 switch is used to create a backup file with an *.BAC extension. The*.BAC files can be reviewed to see if the model is correctly reading the data. This is a data file format troubleshooting tool. These files can be renamed to *.DAT and the model can be run with them. IBACKUP = 1 will also 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 which can be renamed to the FPLAIN.DAT file to run the model.

11. When DEPRESSDEPTH variable is set, every grid element elevation is checked against its neighbors' elevations to see if it represents an isolated depression. A value of DEPRESSDEPTH = 3.0 ft is suggested.

12. DEPRESSDEPTH is also used to identify levees that have a crest elevation tat does not represent reality. Having a levee represent a wall that is only 0.1 ft above the ground is pointless. This condition could occur because of shape file elevation assignments or levee crest interpolation. DEPRESSDEPTH is used to evaluate the minimum difference in the levee crest elevations compared to the ground elevation on both sides of the levee. Two separate independent simulations are required for different DEPRESSDEPTH values (use SIMUL = 0.1 or 0.01 hrs for each). If the variable was used to generate the DEPRESSED_ELEMENTS.OUT file, rename that file and reassign that variable to 1 ft (or lower) to generate LOW_LEVEE_CREST_ELEVATIONS.OUT file.

# FILE: TOLER.DAT

## NUMERICAL STABILITY CONTROL DATA

### TOLER.DAT File Variables

| | |
|---|---|
| 0.1 0.00 | Line 1: **TOLGLOBAL DEPTOL** |
| C 0.6 0.6 0.6 | **Line 2: COURCHAR = 'C' COURANTFP COURANTC ... COURANTST** |
| T 0.1 | **Line 3: COURCHAR = "T" TIME_ACCEL** |

### TOLER.DAT File Example

0.1 0.00
C 0.6 0.6 0.6
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 | **r** | 0.2 - 0.9 | Courant number for channels. Courant-Friedrich-Lewy numerical stability parameter that relates the floodwave movement in channels to the discretized model in space and time (see comments 3 thru 5). |
| COURANTFP | **r** | 0.2 - 0.9 | Courant number for floodplain. Courant number for floodplain. Numerical stability parameter that relates the floodwave movement for overland flow to the discretized model in space and time (see comments 3 thru 5). |
| COURANTST | **r** | 0.2 - 0.9 | Courant number for streets. Courant number for floodplain. Numerical stability parameter that relates the floodwave movement in streets to the discretized model in space and time. (see comments 3 thru 5). |
| COURCHAR | **c** | C, T | Character 'C' that identifies Line 2 with the Courant stability parameter. This variable is case sensitive. It must be upper case. |
| DEPTOL | **r** | 0.1 - 0.5 | Tolerance value for the percent change in the flow depth for a given timestep. When a given element DEPTOL is exceeded, the timestep will be reduced. If DEPTOL = 0, then the timestep is governed by the Courant numerical stability criteria. It is recommended that DEPTOL only be used for specific ponded flow conditions where the Courant number is ineffective (see comment 2). |
| TIME_ACCEL | **r** | 0.1 to 2 | Coefficient to increase the rate of incremental timestep change. Default value = 0.1 A value of 0.1 may result in a a more stable simulation time. A value of 0.2 or higher may result in a faster simulation. |
| TOLGLOBAL | **r** | 0.004 - 0.5 typ 0.0012 - 0.03 | Surface detention. TOLGLOBAL is a minimum value of the flow depth for flood routing. A typical value river flooding is 0.10 ft (see comment 1). Use a small value for rainfall runoff (0.004 ft to 0.10 ft; 0.0012 m to 0.030m). |

# Instructional Comments for the TOLER.DAT File

1. The TOLGLOBAL value prescribes the flow depth for a floodplain or channel grid element below which no flood routing will be performed. TOLGLOBAL is analogous to a depression storage rainfall abstraction. The TOLGLOBAL value for streets is hardwired (0.03 ft or 0.01 m). TOLSPATIAL is another variable that can be assigned to any cell. The TOLSPATIAL variable will replace TOLGLOBAL if assigned. See TOLSPATIAL tab for futher instructions.

2. DEPTOL controls the percent change in grid element or 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. The Courant is the primary numerical stability control. For some models with ponded flow, the water surface and velocities for low n-value may exhibit numerical instability. Using or decreasing DEPTOL will reduce the timestep and, improve the numerical stability and result in longer computational times. Setting DEPTOL = 0 dictates that only the Courant criteria will be applied for numerical stability.

3. To identify numerical instability, review the CHANMAX.OUT file and the HYDROG program hydrograph plots for hydrograph spikes. Review MAXPLOT or Mapper or the VELTIMEFP.OUT file to determine if floodplain velocities are too high.

4. If the model is unstable, reduce the appropriate Courant number by 0.1 in successive runs until the Courant number reaches 0.2.

5. Using the Courant criteria, the timestep $\Delta t$ is limited by:

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

where:

C is the Courant number ($C \leq 1.0$)
$\Delta x$ is the square grid element width
V is the computed average cross section velocity
$\beta$ is a coefficient (e.g. 5/3 for a wide channel) but is seldom used
c is the computed wave celerity

The Courant coefficient C may vary from 0.2 to 0.9 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. Start with the default value of 0.6.

Use the following approach to improve numerical stability:

- Initially run the model with the Courant numbers = 0.6. If the model is unstable, reduce the appropriate Courant number by 0.1 increments in successive runs until the Courant number reaches 0.2.
- 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) and the maximum Froude numbers in SUPER.OUT 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 MANNINGS_N.DAT. Make n-value adjustments in MANNINGS_N.DAT based on exceedingly high n-values in ROUGH.OUT then replace MANNINGS_NDAT with MANNINGS.RGH.
- Run the simulation and repeat steps 3 and 4 making adjustments to MANNINGS_N.DAT until ROUGH.OUT is essentially empty. A few incremental n-value changes will not affect the simulation. Make adjustments to FROUDL to decrease the number of n-value adjustments.

6. Increase the model speed:

- Increase the Courant numbers in 0.1 increments until C = 0.9.
- Increase the TIME_ACCEL parameter in TOLER.DAT in 0.1 increments to increase the computational timesteps increments.
- Review the model numerical stability with the maximum velocity and Froude number output files. Decrease the TIME_ACCEL parameter if unreasonable increases in the maximum velocity and Froude number are reported.
- Review the computational runtime in the SUMMARY.OUT file and balance the increased Courant numbers and TIME_ACCEL parameter to achieve the best runtime. This may require only an increase in TIME_ACCEL.

# 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** | $\infty$ | Ground surface elevation for grid element I (ft or m). |

IMPORTANT NOTE: The FPLAIN.DAT is obsolete and is no longer required if the file NEIGHBORS.DAT exists. It may still appear in the project folder but is no longer necessary to run the model.

FLOPRO.EXE reads the grid, elevation, and Manning's n data as follows:
The model verifies the following files:

TOPO.DAT, FPLAIN.DAT, CADPTS.DAT, NEIGHBORS.DAT, MANNINGS_N.DAT

- If TOPO exists, the model reads it to count the number of grid elements and grid element size.
- If NEIGHBORS.DAT exists, the model reads this file to define the neighbors. If it does not exist, FLOPRO uses TOPO.DAT to define the neighbors and creates NEIGHBORS.DAT. The model starts faster when the file is present.
- If MANNINGS_N.DAT exists, the model reads it to define the floodplain roughness. If the file does not exist but all others do, the model will generate a fatal error message and stop.
- If CADPTS.DAT and FPLAIN.DAT do not exist, the model will generate them.
- If TOPO.DAT and MANNINGS_N.DAT do not exist, the model will use FPLAIN.DAT and CADPTS.DAT to create them.

# Instructional Comments for the FPLAIN.DAT File

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:

$$n_d = n_b * 1.5 * e^{-(0.4\ depth/dmax)}$$

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 or QGIS 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 or QGIS pre-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: This series of files is automatically generated by the FLOPro 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 the TOPO.DAT and MANNINGS_N.DAT files or the 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 is therefore 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: XCOORD(I), YCOORD(I) 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 or QGIS. This files contains the same data as the FPLAIN.DAT and CADPTS.DAT files except for the neighbor grid elements and n-value. It is automatically generated and edited by the GDS or QGIS when the FPLAIN.DAT is written. Use this file together with Mannings_N.DAT for 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 TOPO.DAT data as that contained in FPLAIN.DAT and CADPTS.DAT and is in a format that enables GIS and CADD applications to use it directly. TOPO.DAT has the format of x- and y-coordinate, and elevation (x,y,z file) of the center of the node in a GIS or CADD compatible format.
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. If the TOPO.DAT file is missing at runtime, the model automatically generates it. Conversely if FPLAIN.DAT is missing at runtime, the model automatically generates this file. FPLAIN. DAT is obsolete and is no longer required to run the model

# 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)**<br>*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 50.00 | Line 3: | **HYDCHAR = 'H' HP(J,1) HP(J,2) HP(J,3) J=2** |
| H 24 1553.0 | Line 3: | **HYDCHAR = 'H' HP(J,1) HP(J,2) HP(J,3) J=3** |
| | | OPTIONAL RESERVOIRN(J) |
| R 5232 3320 0.250 | Line 4: | **RESCHAR = 'R' IRESGRID(J) RESERVOIREL(J) RESERVOIRN J=1** |
| R 6528 3295 | Line 4: | **RESCHAR = 'R' IRESGRID(J) RESERVOIREL(J) J=2**<br>*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.
Line 4: Reservoir n value is optional.

### INFLOW.DAT File Example

```
0   4335
C        0    4335
H        0      0
H        1     55.30
H        2    155.30
H        3    253.78
H        4    537.8
H        5    522.7
H        6    507.5
H        7    492.4
R     5232   1734.02
    ....
```

# 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 cms) corresponding to the time interval which starts at HP(I,J,1). |
| HP(I,J,3) | **r** | **0 - 1** | Sediment concentration by volume or sediment volume for a mudflow simulation (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'. Variable is case sensitive. It must be upper case. |
| 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).<br><br>Variable is Case Sensitive. It must be upper case. |
| INOUTFC(I) | **s** | **0 = inflow<br>1 = outflow** | INOUTFC = 0 for inflow hydrograph. INOUTFC = 1 for outflow hydrograph or diversion from the channel (see Comment 4). |
| IHOURDAILY | **s** | **0 = hourly<br>1 = daily** | IHOURDAILY = 0 for inflow hydrograph hourly intervals HP (I,J,1). IHOURDAILY = 1 for daily (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.<br><br>Variable is Case Sensitive. It must be upper case. |

# Variable Descriptions for the INFLOW.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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 - ∞**<br><br>**0 - (- ∞)** | Water surface elevation (ft or m) of the reservoir or ponded water area.<br><br>Negative water surface elevation assigns the reservoir bed elevations below the breach foundation elevation as dead pool ground and reduces the reservoir starting flow depth for those dead pool elements. |
| RESERVOIRN | r | **0.1 - 0.4** | Optional reservoir n-value for all reservoir elements assigned a starting water surface elevation. If RESERVOIRN is not assigned, the model will use the floodplain element n- value. The n-value should be high enough to reduce reservoir velocities to less than 2.0 fps (0.67 mps). A value of 0.25 is suggested (see Comment 6). |

# 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 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 model runtime, the model will automatically assign the same water surface to all the grid elements in an expanding circle of elements around IRESGRID that a have a ground elevation less than the prescribed water surface elevation RESERVOIREL

6. Flooding routing a deep reservoir pool is essentially frictionless flow and should not be simulated using a friction slope given by Manning's equation. Frictionless flow cannot be predicted with the full dynamic equation without a friction slope term. In order to apply the revised Manning's equation for ponded flow, it is recommended that a high n-value be used on the order of 0.1 to 0.4. This will result in reservoir velocities of approximately 1 fps (0.3 mps) which will be representing for filling or draining the reservoir when the water surface slope is almost flat. RESERVOIRN is an optional variable.

# 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 repeat | 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 repeat | 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 ICHANNEL = 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 to a downstream grid system.

# 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 |
|---|---|---|---|
| CHDEPTH(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<br>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<br>O1 - O9** | Character line identifier that initializes each line in the data file (see Comment 7). Variable is case sensitive. It must be upper case. |
| STA_TIME(J) | **r** | **0.0 - ∞<br>500 pairs** | Array of time intervals (hrs) for the grid element stage-time relationship. |
| STA_STAGE(J) | **r** | **0.0 - ∞<br>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 off 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 and enables an approximation of normal flow depth in the outflow elements. This is a simple method to ensure that backwater related to artificial 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 storm surge water surface elevations.

5. If coastal flooding (storm surges or tsunamis) is being simulated with a time-stage hydraulic control, 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 1 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**<br>**1 = on** | IRAINARF = 1 indicates that individual grid element depth-area reduction values will be assigned. |
| IRAINBUILDING | **s** | **0 = off**<br>**1 = on** | IRAINBUILDING = 1 indicates that rainfall on an ARF = 1 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:<br>1 = N 5 = NE<br>2 = E 6 = SE<br>3 = S 7 = SW<br>4 = W 8 = NW |
| IRAINREAL | **s** | **0 = off**<br>**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. |
| MOVINGSTORM | **s** | **0 = off**<br>**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 (see comment 4). |
| RAINCHAR | **c** | **R** | Character 'R' that identifies Line 3. Variable is case sensitive and it must be upper case. |
| RAINSPEED | **r** | **0 - 100**<br>**0 - 50** | Storm speed (mph or kph) |
| RTT | **r** | **0.0 -** $\infty$ | Total storm rainfall (inches or mm). |
| R_TIME(I) | | **0.0 -** $\infty$ | Time (hrs) 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. The rainfall distribution has to be correlated to the flood simulation time. The rainfall may occur for only a portion of the total flood simulation and may start after the flood simulation begins. 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. Similarly the final cumulative rainfall at the end of the simulation could be set equal to 1.0.

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 complete storage loss assigned (ARF = 1 value), the model removes that rainfall 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 building 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.

4. RAINARF values are used for design storm data. The variable is a percentage of the total depth for the cell or the total depth for the cell when using a design storm event in the RAIN.DAT file. For example, set the variable to zero, no rain will fall on the cell. Set it to 0.5, half of the assigned rainfall on that element will be computed for that interval and set the RAINARF value to 1 and all of the rain will fall on the cell. The realtime rainfall (spatially and temporally variable) is also reduced by the RAINARF value over each rainfall interval.

# FILE: RAINCELL.DAT
## REALTIME RAINFALL DATA

### RAINCELL.DAT File Variables

```
15   96  1/1/2000     12:00:00 AM   1/2/2000     12:00:00 AM
                                    Line 1:   RAININTIME   IRINTERS   TIMESTAMP
1   0.0                             Line 2:   IRAINDUM (I)   RRGRID(I,K)
```

### RAINCELL.DAT File Example

```
1          73          4/17/2013     12:00:00 AM   4/20/2013     2:00:00 AM
1          0.0
2          0.0
3          0.0
4          0.0
5          0.0
```

# Variable Descriptions for the RAINCELL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IRAINDUM(I) | **r** | **i - NNOD** | Repeated set of grid elements for each interval. |
| IRINTERS | **r** | **0.0 -** $\infty$ | Number of intervals in the dataset. There will be a complete set of cell values and rain data repeated for each interval. |
| RAININTIME | **r** | **0.0 -** $\infty$ | Time interval in minutes of the realtime rainfall data. This is a single variable in line 1. The time interval starts at zero when the simulation starts. |
| RRGRID(I,K) | **i** | **0.0 -** $\infty$ | Cumulative rainfall in inches or mm over the time interval. |
| TIMESTAMP | **c** | **Alpha Numeric** | Timestamp indicates the start and end time of the storm. (see comment 3) |

# Instructional Comments for the RAINCELL.DAT File

1. Realtime rainfall is spatially and temporally variable rainfall data that is applied to each cell of the grid system. A complete dataset is stored in the data file RAINCELL.DAT. The rainfall time interval is repeated on the minute so data is commonly compiled in fifteen minute intervals.

2. The GDS and QGIS can interpolate this file from a series of ASCII grid files. Each file will contain the data for a single interval. The files are organized using a catalog. The catalog is loaded into the GDS or QGIS. It is a simple text file as such: raincatalog.rtc. A small sample of the catalog data is shown below.

```
7/13/2008 10:00 7/13/2008 15:00 1 5
C:\Projects\NexRAD\Min1.asc
C:\Projects\NexRAD\Min2.asc
C:\Projects\NexRAD\Min3.asc
C:\Projects\NexRAD\Min4.asc
C:\Projects\NexRAD\Min5.asc
```

3. The timestamp is not used by the GDS, QGIS or FLOPRO.EXE engine. It is a reference variable. It can be used to synchronize the raincell storm data to inflow hydrographs.

# 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)**<br>*IC= number of channel segments or reaches* |
| 99 0 | Line 5: | **SCSNALL ABSTR1** |
| F 1730 0.01 4.3 0.3 0.0 0.0 10.0 | Line 6: | **INFILCHAR = 'F' INFGRID(IF) HYDC(IF) SOILS(IF) DTHETA(IF) ABSTRINF(IF) RTIMPF(IF) SOIL_DEPTH(IF)**<br>*IF = 1 - number of infiltration data sets* |
| S 320 82.00 | Line 7: | **INFILCHAR = 'S' INFGRID(IF) SCSCN(IF)** |
| 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 3450 3.0 0.5 0.00018 | Line 10: | **INFILCHAR = 'H' INFGRID(IF) FHORTI(INFGRID(IF)) FHORTF(INFGRID(IF)) DECA(INFGRID(IF))**<br>*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

## 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**<br>**0 - 25** | Green Ampt global floodplain rainfall abstraction or interception (inches or mm) (see comments 1 and 7). |
| ABSTRINF(N) | **r** | **0 - 1**<br>**0 - 25** | Grid element rainfall abstraction (inches or mm). |
| ABSTR1 | **r** | **0 - 1**<br>**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<br>(INFGRID(IF)) | **r** | **0.0007 - 0.0018** | Horton's equation spatially variable decay coefficient (1/second; no metric) (see comment 14). |
| DECAYA | **r** | **0.0007 - 0.0018** | Horton's equation decay coefficient (1/second; no metric) (see comment 14). |
| DTHETA(N) | **r** | **0.0 - 1**<br>**0.0 - 0.5** | The 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 grid element volumetric soil moisture deficit that is defined as the soil moisture deficit multiplied by the porosity (SATF-SATI)*POROS with a range from 0.3 to 0.5 (see comment 11). Set POROS = 0 for the volumetric soil moisture deficiency. |
| DTHETAC(I) | **r** | **0.0 - 1**<br>**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 range from 0.3 to 0.5 (see comment 11). Set POROS = 0 for the volumetric soil moisture deficiency. |
| FHORTF<br>(INFGRID(IF)) | **r** | **0.5 - 1.0** | Horton's equation spatially variable floodplain final infiltration rate (inches/hr; no metric). |
| FHORTI<br>(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 14). |
| HYDC(N) | **r** | **0.01 - 10**<br>**0.25 - 250** | Grid element average hydraulic conductivity of an (inches/hr or mm/hr) (see comments 2, 4 and 5). |
| HYDCALL | **r** | **0.01 - 10**<br>**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:<br>If HYDCADJ < 1.00 HYDCS(i) = HYDCS(i) + HYDCADJ<br>If HYDCADJ > 1.00 HYDCS(i) = HYDCS(i) * HYDCADJ |
| HYDCHN | **r** | **0.01 - 10**<br>**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**<br>**0.25 - 250** | Channel reach hydraulic conductivity channel (inches/hr or mm/hr) (see comment 8). |
| HYDCONCH(N) | **r** | **0.01 - 10**<br>**0.25 - 250** | Hydraulic conductivity for a channel element (inches/hr or mm/hr). |
| HYDCX(IC) | **r** | **0.01 - 10**<br>**0.25 - 250** | Initial hydraulic conductivity for a channel segment (inches/hr or mm/hr) (see comment 15). |
| HYDCXFINAL(IC) | **r** | **0.01 - 10**<br>**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**<br>**1 = on** | Set switch to 1 to simulate channel infiltration (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’ = spatially variable floodplain Green-Ampt data (Line 6),<br>‘S’ = floodplain spatially variable SCS curve number (Line 7);<br>‘C’ = channel spatially variable channel infiltration (Line 8);<br>‘R’ = channel reach infiltration data (Line 4 and 4a);<br>‘I’ = Horton global parameters (Line 9);<br>‘H’ = Horton spatially variable floodplain data (Line 10).<br><br>Variable is case sensitive and it must be upper case. |
| INFMETHOD | **s** | **1, 2, 3 or 4** | 1: Green-Ampt method;<br>2: SCS curve number method;<br>3: Combined Green-Ampt and CN methods;<br>4: Horton method. |
| 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 (decimal percentage) for computing infiltration. |
| SATI | **r** | **0.0 - 0.95** | Global initial saturation of the soil (decimal percentage). |
| 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 12). |

# 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**<br>**25 - 500** | Capillary suction head for floodplain grid elements (inches or mm). |
| SOILALL | **r** | **1 - 20**<br>**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. Floodplain grid elements with unique Green-Ampt infiltration parameters are specified in Line 6 which supersede then global 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 runoff, however, is assumed to occur for an ARF = 1 grid element if IRAINBUILDING = 1 in the RAIN.DAT file. 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 TOL value of ranging from 0.004 to 0.1 ft (0.001 to 0.03 m) represents the depression storage.
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 SCS curve number method (INFMETHOD = 2) is used, it is assumed that the channel infiltration is negligible. Simulate channel infiltration with the Green-Ampt method.

10. 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 in Line 1 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 seepage or bank infiltration.

14. 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.

15. 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 soil 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 project.

### 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
   ...
```

# Variable Descriptions for the EVAPOR.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| CLOCKTIME | **r** | **0.0 - 24.0** | Starting clock time (hrs) of the simulation time during the day. |
| EMONTH(I) | **c** | **Jan - Dec** | Name of month for user identification purposes only. |
| EVAP(I) | **r** | **0 - 100**<br>**0 - 2500** | Monthly evaporation rate (in/month or mm/month). |
| EVAPER(I,J) | **r** | **0.0 - 1.0** | Hourly percentage of the daily total evaporation for each month. There will be 24 values that will total 1.00 for each of the twelve months. |
| IEVAPMONTH | **i** | **1 - 12** | Starting month of simulation. |
| IDAY | **i** | **1 - 7** | Starting day of the week. |

# FILE: CHAN.DAT

## CHANNEL DATA

### CHAN.DAT File Variables

Line 1: **DEPINITIAL(K) FROUDC(K) ROUGHADJ ISEDN(K)**

0.00 0.50 0.20 0

Line 2a: **SHAPE** *'R' = Rectangular* **ICHANGRID(I) BANKELL(I) BANKELR(I) FCN(I) FCW(I) FCD(I) XLEN(I)**

R 50 4765.52 4765.00 0.031 22.54 6.32 100.00

Line 2b: **SHAPE** *'V' = Variable Area* **ICHANGRID(I) BANKELL(I) BANKELR(I) FCN(I) FCD(I) XLEN(I) A1(I) A2(I) B1(I) B2(I) C1(I) C2(I) EXCDEP(I) A11(I) A22(I) B11(I) B22(I) C11(I) C22(I)**

V 50 4765.52 4765.00 0.031 6.32 505.00 36.77 1.63 63.37 0.491 63.261 0.49 0.00

Line 2c: **SHAPE** *'T' = Trapezoidal* **ICHANGRID(I) BANKELL(I) BANKELR(I) FCN(I) FCW(I) FCD(I) XLEN(I) ZL(I) ZR(I)**

T 50 4765.52 4765.00 0.031 22.54 6.32 100.00 2.40 1.50

Line 2d: **SHAPE** *'N' = Natural* **ICHANGRID(I) FCN(I) XLEN(I) NXECNUM(I)**

N 50 1 0.031 100.00 1

| | |
|---|---|
| 50 4763.00 | Line 3a: **ISTART WSELSTART** |
| 77 4761.00 | Line 3b: **IEND WSELEND** |
| C 501 498 | Line 4: **CHANCHAR = 'C' ICONFLO1(J) ICONFLO2(J)** |

*I = number of channel nodes.*
*J = number of channel confluences*
*K = number of channel segments*
*N = number of nofloc and noexchange data sets*

Notes:
If ICHANNEL = 0 in the CONT.DAT file, omit this file.
Line 1: This line is repeated at the start of each channel segment. If ISED = 0 in the CONT.DAT file omit ISEDN(K).
Line 2: This line is repeated for each channel grid element. Use 2a, 2b, 2c, or 2d for this line.
Line 3: If not simulating an initial water surface elevation in the channel, omit this line. Repeat 3a and 3b for each channel segment.
Line 3, 4 and 5: Multiple lines are grouped together.

# 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  ...
```

# 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 - ∞** | Coefficient for the variable area regression relationships (see comment 5). |
| A2(I) | **r** | **0.0 - ∞** | Exponent for the variable area regression relationships (see comment 5). |
| A11(I) | **r** | **0.0 - ∞** | Coefficient for the variable area regression relationships for flow depth above EXCDEP(I) (see comment 5). |
| A22(I) | **r** | **0.0 - ∞** | Exponent for the variable area regression relationships for flow depth above EXCDEP(I) (see comment 5). |
| B1(I) | **r** | **0.0 - ∞** | Coefficient for the variable wetted perimeter relationships (see comment 5). |
| B2(I) | **r** | **0.0 - ∞** | Exponent for the variable wetted perimeter relationships (see comment 5). |
| B11(I) | **r** | **0.0 - ∞** | Coefficient for the variable wetted perimeter relationships for flow above EXCDEP(I) (see comment 5). |
| B22(I) | **r** | **0.0 - ∞** | Exponent 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 - ∞** | Coefficient for the variable top width relationships (see comment 5). |
| C2(I) | **r** | **0.0 - ∞** | Exponent for the variable top width relationships (see comment 5). |
| C11(I) | **r** | **0.0 - ∞** | Coefficient for the variable top width relationships for flow depth above EXCDEP(I) (see comment 5). |
| C22(I) | **r** | **0.0 - ∞** | Exponent 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. Variable is case sensitive and it must be upper case. |
| ICONFLO1(J) | **i** | **1 - NNOD** | Tributary channel element at confluence (see comment 8). |

# Variable Descriptions for the CHAN.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| ICONFLO2(J) | **i** | **1 - NNOD** | Main channel element at the confluence. |
| DEPINITIAL(K) | **r** | **0.0 - ∞ or -1** | DEPINITIAL(K) = 0 for no initial channel flow depth in the channel segment (default).<br>DEPINITIAL(K) = Initial flow depth for the all channel elements in the channel segment (optional).<br>DEPINITIAL(K) = -1 to assign an initial water surface elevation to a channel reach. Include Line 3 (see comment 2). |
| EXCDEP(I) | **r** | **0.0 - ∞** | Channel depth 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 - 0.15** | Average Manning's n roughness coefficient the channel in the grid element ICHANGRID (see comments 6 and 19). |
| FCD(I) | **r** | **.01 - 1000** | Channel thalweg depth (ft or m). 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.<br>Set FCW(I) = width of channel base for trapezoidal channel. |
| FROUDC(K) | **r** | **0.0 - 5** | Maximum channel Froude number 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 channel segment. The increased n-values are reported in the ROUGH.OUT and CHAN.RGH files (see comment 7). |
| ICHANGRID(I) | **i** | **1 - NNOD** | Channel grid element number. |
| 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:<br>For sediment routing without size fractions: Set ISEDN(K) = 1 - 11 (one of eleven sediment transport equations).<br>or<br>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. |
| 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 is 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 and the shallown value for channel segments (see comment 17). |
| SHAPE | **c** | **R, V, T or N** | Character line identifier (see comments 4 and 16);<br>SHAPE = 'R', rectangular channel geometry (width and depth data).<br>SHAPE = 'V', variable area channel geometry (power relationships).<br>SHAPE = 'T', trapezoidal channel (bottom width, depth and slopes data).<br>SHAPE = 'N', channel cross sections (cross section survey data).<br>Variable is case sensitive and it must be upper case. |
| WSELEND | **r** | **0 - 30,000<br>0 - 9,000** | Ending water surface elevation for the channel element IEND (ft or m). |
| WSELSTART | **r** | **0 - 30,000<br>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 - ∞** | 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 100 ft wide can be used in a 20 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 bank 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. The confluence can be made by the tributary joining 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 surveyed 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 if 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 improve the numerical stability around the inflow and outflow nodes.

14. After deleting a channel element, remove the cross section for that channel element from the XSEC.DAT file and renumbered in the PROFILES program. If cross sections are mixed with other channel geometry (trapezoidal or rectangular), 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. If the channel routing is unstable or numerically surging, reduce the Courant number C in the TOLER. DAT by 0.1.

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 ROUGHADJ value (from 0 to 1.2) as 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.

A channel spatially variable shallow n-value assigned to the depths less than 0.2 ft (0.067 m) is defined by applying the ROUGHADJ to each channel reach.:

SHALLOWN = ROUGHADJ / 2

where: ROUGHADJ is assigned to line 1 of each channel segment.

18. Instructions for creating the cross section channel geometry data files are outlined in Lesson 14 of the Workshop Lessons. The lessons are found in the FLO-2D Pro Documentation folder.

19. Surveyed water surfaces can be automatically compared with the predicted water surface in the PROFILES program 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. See comments 1 to 3. |
| RIGHTBANK | **i** | **1 - NNOD** | Right bank channel grid element corresponding the LEFT-BANK element. |

# Instructional Comments for the CHANBANK.DAT File

1. The RIGHTBANK element is automatically assigned in the GDS or QGIS. Make adjustments to the right bank channel element if the channel is too wide or narrow by reassigning the right bank element in the GDS or QGIS. It is also assigned if unequal channel bank elevations are assigned in CHAN.DAT regardless if the channel will fit into one grid element.

2. The procedure for assigning the right bank element is to first select the left bank element in the GDS or QGIS, open the channel segment editor box and then assign the right bank element. The GDS or QGIS 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, .. n number of cross sections** |
| 25.0 5234.90 | Line 2: **XI(I,J) YI(I,J)** |
| 30.0 5231.53 | Line 2: **XI(I,J) YI(I,J)** |
| 35.0 5230.20 | Line 2: **XI(I,J) YI(I,J)** ... ........................................................... |

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
       ...              ...
```

# 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 - ∞** | Cross section station distance from the left end point (ft or m). The value of XI can be either positive or negative. |
| XSECHAR | **c** | **X** | Character 'X' that identifies Line 1. Variable is case sensitive and it must be upper case. |
| XSECNAME(I) | **c** | **Alpha Numeric** | Cross section name (less than 15 characters, not case sensitive). This name is for cross section ID purposes only and it is not used by the model.. Do not use spaces in the name. |
| YI(I,J) | **r** | **0 - 30,000**<br>**0 - 9,000** | Cross section elevation (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. Use the GDS, QGIS or PROFILES programs to interpolate a cross section to each channel element.

# 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*

Line 2: **STRUCHAR = 'B' IBTYPE(I) COEFFP(I) C_PRIME_USER(I) KF_COEF(I) KWW_COEF(I) KPHI_COEF(I) KY_COEF(I) KX_COEF(I) KJ_COEF(I)** *I = number of bridges in bridge routine*

Line 3: **STRUCHAR = 'B' BLENGTH(I) BN_VALUE(I) UPLENGTH12(I) LOWCHORD(I) DECKHT(I) DECKLENGTH(I) PIERWIDTH(I) SLUICECOEFADJ(I) ORIFICECOEFADJ(I) COEFFWEIRB(I) WING WALL_ANGLE(I) PHI_ANGLE(I) LBTOEABUT(I) RBTOEABUT(I)** *I = number of bridges in bridge routine*

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

# FILE: HYSTRUC.DAT
## HYDRAULIC STRUCTURE DATA

### HYSTRUC.DAT File Notes

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: For generalized culverts (ICURVTABLE(I) = 2), 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 discharge pair.
Line 1, 4: If CLENGTH(I) = 0, ignore ATABLE(I,J).

### 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
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 - ∞** | When the long culvert routine is used (CLENGTH(I,J) > 1) must be included as data input. QTABLEA(I,J) is the hydraulic structure flow area for each headwater depth in the rating table (discharge). |
| COEFA(I,J) | **r** | **0 - ∞** | When the long culvert routine is used (CLENGTH(I,J) > 1),. COEFQ(I,J) is the flow area rating curve coefficient where the flow area A is expressed as a power function of the headwater depth. A = COEFA(I,J) * $depth^{EXPA}$(I,J). |
| COEFQ(I,J) | **r** | **0 - ∞** | Discharge rating curve coefficients 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 culvert equations CDIAMETER is the circular culvert diameter or the box culvert height (see comment 12). |
| CLENGTH(I,J) | **r** | **1 - ∞**<br>**1 - ∞** | Culvert length (ft or m). When a long culvert is simulated (>300 ft or 100 m), CLENGTH must be assigned to that culvert's length to activate the long culvert routing 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 - ∞** | When the long culvert routine is used (CLENGTH(I,J) > 1), EXPQ(I,J) is the hydraulic structure flow area exponent where the flow area is expressed as a power function of the headwater depth. |
| EXPQ(I,J) | **r** | **0 - ∞** | Hydraulic structure discharge exponent where the discharge is expressed as a power function of the headwater depth. |
| HDEPEXC (I,J) | **r** | **.01 - 1000**<br>**0.25 - 300** | Maximum depth that a given hydraulic structure rating curve is valid (ft or m). |
| HDEPTH(I,J) | **r** | **.01 - 1000**<br>**0.25 - 300** | Headwater depth for the structure headwater depth-discharge rating table (ft or m) (see comment 2). |

# 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<br>.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<br>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 5). |
| IFPORCHAN(I) | **s** | **0, 1, 2 or 3** | IFPORCHAN(I) = 0; for a floodplain structure (shares discharge between two floodplain elements).<br>IFPORCHAN(I) = 1; for a channel hydraulic structure (shares discharge between two channel elements).<br>IFPORCHAN(I) = 2; for a floodplain to channel structure (shares discharge between a floodplain element {inflow} and channel structure {outflow})(see comment 7).<br>IFPORCHAN(I) = 3; for a channel to floodplain structure (shares discharge between a channel {inflow} element and a floodplain element {outflow}) (see comment 13). |
| INFLONOD(I) | **i** | **1 - NNOD** | Grid element containing the hydraulic structure or structure inlet. |
| INOUTCONT(I,J) | **s** | **0 = inlet<br>1 = revised<br>2 = outlet / revised** | INOUTCONT(I,J) = 0; to compute the discharge based on only the headwater depth above the appropriate floodplain or channel bed elevation (or reference elevation if assigned). Suggested revisions are listed in REVISED_RATING_TABLE.OUT. No tailwater effects or potential upstream flow are considered.<br>INOUTCONT(I,J) = 1; reduced discharge for tailwater submergence, , but does not allow upstream flow. Suggested rating table revisions posted to REVISED_RATING_ TABLE.OUT.<br>INOUTCONT(I,J) = 2; reduced discharge for tailwater submergence. Upstream flow is possible. Suggested rating table revisions posted to REVISED_RATING_TABLE.OUT.<br>Always set to 0 for Generalized Culvert Equation. (see comment 15). |

# Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| ISTORMDOUT(I) | **i** | **1 - NNOD** | Hydraulic structure outflow grid element number used to simulate a simplified storm drain. ISTORMDOUT is a junction or outflow node for a number of inflow nodes (see comment 11). |
| KE(I) | **r** | **0.01 - 1.0** | Culvert entrance loss coefficient (see comment 9). |
| 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 - ∞** | Hydraulic structure discharges for the headwater depths in the rating table (discharge) (see comments 3 and 4). |
| 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 - ∞** | When the long culvert routine is used (CLENGTH(I,J) > 1), RACOEF(I,J) is the structure rating curve flow area replacement coefficient. There should be the same number of rating curve pairs of coefficients and exponents. |
| RAEXP(IJ) | **r** | **0 - ∞** | When the long culvert routine is used (CLENGTH(I,J) > 1), RAEXP(I,J) is the structure rating curve flow area 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:<br><br>STRUCHAR = 'S' for the structure control, (Line 1);<br>STRUCHAR = 'C' for a rating curve (Line 2);<br>STRUCHAR = 'R' for replacement rating curve (Line 3);<br>STRUCHAR = 'T' for a rating table (Line 4);<br>STRUCHAR = 'F' for culvert equations (Line 5);<br>STRUCHAR = 'D' for storm drain (Line 6).<br><br>Variable is case sensitive and it must be upper case. |

# Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| STORMDMAXQ(I) | **r** | **0 - ∞** | Maximum allowable discharge (conveyance capacity) of the collection pipe represented by the ISTORMDOUT element. |
| STRUCTNAME(I) | **c** | **Alpha Numeric** | Hydraulic structure name (15 characters or less). This name is for user identification purposes only. No spaces allowed in the name. |
| 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 8). |
| TYPEEN(I) | **s** | **1, 2, 3** | Culvert switch. Set TYPEEN(I) for entrance type 1, 2, or 3. (see comment 8). |
| | | | |
| STRUCHAR | **c** | **B** | Character identifier for the bridge routine (see comment 16). |
| IBTYPE | **i** | **1 - 4** | Type of bridge configuration (see White Paper graphics) |
| COEFF* | **r** | **0.1 - 1.0** | Overall bridge discharge coefficient – assigned or computed (default = 0.). See comment 17. |
| C_PRIME_USER* | **r** | **0.5 - 1.0** | Baseline bridge discharge coefficient to be adjusted with detail coefficients |
| KF_COEF* | **r** | **0.9 - 1.1** | Froude number coefficient – assigned or computed (= 0.) |
| KWW_COEF* | **r** | **1.0 - 1.13** | Wingwall coefficient – assigned or computed (= 0.) |
| KPHI_COEF* | **r** | **0.7 - 1.0** | Flow angle with bridge coefficient – assigned or computed (= 0.) |
| KY_COEF* | **r** | **0.85 - 1.0** | Coefficient associated with sloping embankments and vertical abutments (= 0.) |
| KX_COEF* | **r** | **1.0 - 1.13** | Coefficient associated with sloping abutments – assigned or computed (= 0.) |
| KJ_COEF* | **r** | **0.6 - 1.0** | Coefficient associated with pier and piles – assigned or computer (= 0.) |
| | | | |
| BOPENING | **r** | **0.0 - ∞** | Bridge opening width (ft or m) |
| BLENGTH | **r** | **0.0 - ∞** | Bridge length from upstream edge to downstream abutment (ft or m) |
| BN_VALUE | **r** | **0.030 - 0.200** | Bridge reach n-value |
| UPLENGTH12 | **r** | **0.0 - ∞** | Distance to upstream cross section unaffected by bridge backwater (ft or m) |
| LOWCHORD | **r** | **0.0 - ∞** | Average elevation of the low chord (ft or m). |

# Variable Descriptions for the HYSTRUC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| DECKHT | **r** | **0.0 - ∞** | Average elevation of the top of the deck railing for overtop flow (ft or m) |
| DECKLENGTH | **r** | **0.0 - ∞** | Deck weir length (ft or m) |
| PIERWIDTH | **r** | **0.0 - ∞** | Combined pier or pile cross section width (flow blockage width in ft or m) |
| SLUICECOEFADJ | **r** | **0.0 - 2.0** | Adjustment factor to raise or lower the sluice gate coefficient which is 0.33 for Yu/Z = 1.0. See comment 18. |
| ORIFICECOEF-ADJ | **r** | **0.0 - 2.0** | Adjustment factor to raise or lower the orifice flow coefficient which is 0.80 for Yu/Z = 1.0 |
| COEFFWEIRB | **r** | **2.65 - 3.21** | Weir coefficient for flow over the bridge deck. For metric: COEFFWIERB x 0.552. Comment 19. |
| WINGWALL_ANGLE | **r** | **$30^0$ - $60^0$** | Angle the wingwall makes with the abutment perpendicular to the flow |
| PHI_ANGLE | **r** | **$0^0$ - $45^0$** | Angle the flow makes with the bridge alignment perpendicular to the flow |
| LBTOEABUT | **r** | **ELEVATION** | Toe elevation of the left abutment (ft or m) |
| RBTOEABUT | **r** | **ELEVATION** | Toe elevation of the right abutment (ft or m) |
| * If the coefficient is assigned 1.0, that bridge coefficient is either not important or has no effect. | | | |

# Instructional Comments for the HYSTRUC.DAT File

1. There are two approaches for bridge flow, rating curve/table or computing the bridge flow hydraulics directly as free surface, pressure flow or pressure and weir flow. For the rating curves, the hydraulic structure discharge between either floodplain or channel grid elements can be simulated as a 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 rating table is applied to estimate the discharge for a headwater depth computed by the model.

4. 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.

5. 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 the simulation unless the culvert can contain a significant portion of the flood volume in the entire model.

6. If the culvert is long (CLENGTH > 300 ft or 100 m), Muskingum-Cunge volume routing is applied. The flow area for the culvert is required as given by the variables COEFA, EXPA, RACOEF, RAEXP and ATABLE. The model will

automatically substitute use the long culvert volume routing when CLENGTH > 1.

7. If the hydraulic structure is a bridge, culvert or weir between two floodplain elements, set IFPORCHAN = 0. If the structure in a channel, set IFPORCHAN = 1. If the structure such as a culvert or pump collects discharge from a floodplain and discharges to a channel, set IFPORCHAN = 2. Finally is the hydraulic structure has an inlet in a channel element and an outlet in a floodplain element, IFPORCHAN =3.

8. 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

9. The culvert equations use the conventional entrance loss coefficients KE values that be found in the literature.

10. If INOUTCONT(I,J) = 0, then the hydraulic structure discharge is based solely on the upstream water surface elevation ( headwater depth above the reference elevation which is either assigned or represents the node elevation). This is equivalent to inlet control for a culvert. If INOUTCONT(I,J) = 1, then the tailwater submergence is evaluated. As the tailwater elevation approaches the upstream headwater elevation, the model adjusts the rating curve or table and gradually reduces the discharge as the outlet becomes submerged. When the switch INOUTCONT(I,J) = 2, submergence discharge reduction occurs and if the tailwater elevation exceeds the headwater elevation then flow upstream is possible. When 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 reduces the structure discharge. This may occur because of tailwater effects or because the structure discharge rating table was overestimated for the upstream flow conditions. The submergence modifications to the rating table are reported in the REVISED_RATING_TABLES.OUT file.

11. By assigning ISTORMDOUT, the discharge from this outflow element will represent the collective inflow from any number of upstream inflow elements with the same outflow node. The discharge in the outflow element ISTORMD-

OUT 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 storm drain component. The purpose of this component is to estimate the collected discharge in a large series of culverts or a limited storm drain network. It will limit the potential inflow as the pipe capacity is reached without performing a pipe network discharge calculation. For complex pipe networks, use the FLO-2D storm drain model.

12. CDIAMETER is primarily used to estimate the timing of flow through a long culvert. This is accomplished with a Muskingum-Cunge method of storage routing. When the culvert is longer than about 300 ft (100 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).

13. A hydraulic structure can be set up to compute flow exchange from a channel element to a floodplain node. For example, a channel may share flow through a weir structure to a retention basin represented by floodplain elements.

14. For hydraulic structures simulation of pumps, set the intake elevation as the Head Reference Elevation. The default value is zero. This setting will use the grid element elevation of the inlet node intake elevation for the pump. That may be incorrect and result in a negative head on the intake.

15. For generalized culverts, the INOUTCONT should be set to 0. The tailwater conditions are inherent in the generalized culvert equations for normal depth and critical depth.

16. The bridge hydraulic routine replaces the need for rating curves or rating tables and represents significant added detail in computing free surface flow, pressure flow or combined pressure and weir discharge for flow over the deck. See the White Paper “Bridge Hydraulics Component” for the details on the bridge flow routine that is available in the FLO-2D Help folder.

17. In the bridge hydraulics component, the free surface flow is computed using various coefficients that represent the bridge features impact on the flow as a function of water surface elevation (such as piers). The user can assign the coefficients directly or use the automated interpolation of the USGS coefficients from the White Paper Appendix figures.

18. Bridge pressure flow is computed as either sluice gate flow or orifice flow depending on the water surface elevation with respect to the bridge soffit.

19. When the water surface exceeds the bridge deck elevation, broadcrested weir flow is computed. This is added to the pressure flow to determine the total discharge through the bridge. It is recommended that the weir coefficient be estimated on the low side to account for spaced rails, walkways, debris and other non-uniform deck features.

# 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 directions* |
| W 2 50 | Line 4: | **STRCHAR = 'w', ISTDIR(K) WIDR(K)** *K = 1,8 street directions* |
| W 4 50 | Line 4: | **STRCHAR = 'w', ISTDIR(K) WIDR(K)** *K = 1,8 street directions* |
| 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**<br>**0.0 - 0.6** | Global street curb height (ft or m). If the street curb height is exceeded by the flow it will result in overland flow depth 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**<br>**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**<br>**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 the 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:<br>1 = north 5 = northeast<br>2 = east 6 = southeast<br>3 = south 7 = southwest<br>4 = west 8 = northwest |
| ISTRFLO | **s** | **0 or 1** | ISTRFLO = 1 specifies that the floodplain 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 spatially variable street n-value within a given grid element. STMAN(L) supercedes the STRMAN value. If STMAN(L) = 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. No spaces allowed. (see comment 1). |
| WIDR(K) | **r** | **0.0 - 1,000**<br>**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 elements. The street depth, width, n-value and elevation can be spatially variable for the individual grid elements.

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 over-all 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 occupy all the grid element surface area, consider leaving out the smaller less significant streets, reduce the street width or transfer one or more streets segments to a neighboring grid element. Another option is to increase the grid element size and reassign 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 tolerance value TOLST below which no street flow routing computations are performed is 0.03 ft or 0.01 meters. This value is replaces the floodplain tolerance TOL value in TOLER.DAT and 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**<br>**or**<br>**-1 - 0** | Area reduction factors (ARF) is the percent of the grid element (I) area that cannot be covered by surface flows. Buildings or other physical features within the grid element that cannot store flow volume are accounted by using the ARF value. The maximum ARF value is limited according to cell size (see comments 1 and 3).<br>If the value is negative, the building collapse function is turned on (see comment 5). |
| ARFBLOCKMOD | **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** | Partially blocked grid element numbers. |
| ITTAWF(I) | **i** | **1 - NNOD** | 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. If this value is negative, the building collapse feature is turned on for the entire cell (see comment 5). |
| ITTCHAR | **c** | **S, T** | Set ITTCHAR = 'S' to identify Line 1.<br>Set ITTCHAR = 'T' to identify Line 2.<br>Variable is case sensitive and it must be upper case. |

# Variable Descriptions for the AFR.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| WRF(I,J) | **r** | **0 - 1** | Width reduction factors (WRF). The width reduction factor corresponds to the percentage of flow width blocked due to obstruction in the eight flow directions. Assuming that the flow field is oriented with the north direction, use the following WRF assignment:<br>WRF(I,1) = North<br>WRF(I,2) = East<br>WRF(I,3) = South<br>WRF(I,4) = West<br>WRF(I,5) = Northeast<br>WRF(I,6) = Southeast<br>WRF(I,7) = Southwest<br>WRF(I,8) = Northwest<br>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 relatively easy in the GDS or QGIS programs.

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. If blockage redundancy is written to ARF.DAT, the model will use the more restrictive WRF value.

3. The maximum ARF value is dependent on the grid element size 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. ARF values will be adjusted to prevent numerical instability. The following table lists the adjustment triggers.

**TABLE 4.2. ARF VALUES TRIGGER MAX**

| Grid Element Size | Maximum ARF Reset to 1 |
|---|---|
| Cell Side > 50 | 0.95 |
| 20 < to < 50 | 0.90 |
| 20 > Cell Side | 0.85 |

5. Instead of completely blocking any flow from entering the ITTAWF elements, assigning ARFBLOCKMOD < 1. will allow some flow storage in these completely blocked elements. This variable only modifies totally blocked elements.

6. To assess the potential for building collapse, assign the totally blocked element (-ITTAWF) or the partially blocked ARF value (-ARF) in ARF.DAT as a negative value. Each building element that could collapse must be assigned a negative value. If a building consists of multiple totally blocked elements (ITTAWF - T-line in ARF.DAT), all of the ITTAWF grid element numbers must be assigned as negative to completely remove the building.

# FILE: MULT.DAT
## MULTIPLE CHANNEL (RILL AND GULLY) DATA

### MULT.DAT File Variables

Line 1: **WMC WDRALL DMALL NODCHNSALL XNMULTALL**
0 0.0 5.0 1 0.04 0.00 0.00 0.0 **SSLOPEMIN, SSLOPEMAX AVULD50**

1961 3.0 5.0 1 0.04 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

```
0  0.0  5.0  1  0.04  0.00  0.00  0.0
1961  3.0  5.0  1  0.04
1962  3.0  5.0  1  0.04
1963  3.0  5.0  1  0.04
1964  3.0  5.0  1  0.04
1965  3.0  5.0  1  0.04
1966  3.0  5.0  1  0.04
1967  3.0  5.0  1  0.04
1968  3.0  5.0  1  0.04
1969  3.0  5.0  1  0.04
1970  3.0  5.0  1  0.04
1971  3.0  5.0  1  0.04
```

# Variable Descriptions for the MULT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| AVULD50 | **r** | **0.00 - 100.0** | Bed material D50 sediment size fraction (mm). Assignment of AVULD50 triggers the avulsion component that will allow a multiple channel to seek a new path if the channel conveyance capacity is exceeded (see comment 6). |
| DM(K) | **r** | **0 - 1000**<br>**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 supersedes the DMALL depth assignment. |
| DMALL | **r** | **0 - 1000**<br>**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 supersedes 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**<br>**0 - 300** | Channel width for individual grid elements. WDR supersedes WDRALL. |
| WDRALL | **r** | **0 - 1000**<br>**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**<br>**0 - 300** | Incremental width by which multiple channels will be expanded when the maximum depth DM is exceeded (see comments 2 and 4). |
| XNMULT(K) | **r** | **0.01 - 0.5** | Channel n-values for individual grid elements. Supersedes 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). The program will automatically eliminate any multiple channels in grid elements with streets. The available surface area and the assigned variable can be reviewed in the SURFAREA.OUT output file.

2. If a multiple channel fills and is about to overflow, it is assumed that it is an alluvial channel and will widen to accept more flow. Thus when the flow depth exceeds the maximum channel depth DM, the model increases the width by WMC to maintain the channel conveyance. The multiple channel will not overflow on the floodplain, but will continue to widen until the gully is wider than 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 spatially variable grid element data will supersede the global data.

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 corresponding width.

5. SSLOPEMIN and SSLOPEMAX define a range of watershed slope in which the multiple channel width will be expanded. This will limit the channel width growth to the middle portion of the basin and will not expand the other multiple channels. By expanding the channels for increased conveyance capacity, the time of concentration can be reduced. The default SSLOPEMIN = SSLOPEMAX = 0.0 will result in width increases for all multiple channels.

6. The avulsion component will be initiated if AVULD50 > 0. When a multiple channel conveyance capacity is exceeded in a given grid element, the model will search the other flow direction neighbor elements without a multiple channel

and will create a multiple channel in that grid element based on width and depth relationship as a function of bed material size (AVULD50). This will continue in the downslope direction until the multiple channel conveyance capacity is no longer exceeded. For more information, see the avulsion discussion white paper in the FLO-2D Handouts and Reference Manual.

# 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 a sediment transport ISED or mudflow MUD simulation can be applied in a project model.
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 debris basin 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 exponent below). |
| BEDTHICK | **r** | **0 - 100**<br>**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. $Q_s = ASED^* \, Q_w^{\ BSED}$<br>where:<br>$Q_w$ is the water discharge (cfs or cms)<br>$Q_s$ 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 superseded 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 supersedes 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). It is used only in the Woo-MPM sediment transport equation. |
| 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). |

SED.DAT

# Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| ISEDCFP(N) | **s** | **0 = fp**<br>**1 = chan** | ISEDCFP(N) = 0 for a floodplain sediment supply rating curve<br>ISEDCFP(N) = 1 for a channel sediment supply rating curve. |
| 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:<br>ISEDEQ = 1 Zeller and Fullerton<br>ISEDEQ = 2 Yang<br>ISEDEQ = 3 Englund and Hansen<br>ISEDEQ = 4 Ackers and White<br>ISEDEQ = 5 Laursen<br>ISEDEQ = 6 Toffaleti<br>ISEDEQ = 7 Woo-MPM<br>ISEDEQ = 8 MPM-Smart<br>ISEDEQ = 9 Karim-Kennedy<br>ISEDEQ = 10 Parker, Klingeman & McLean<br>ISEDEQ = 11 Van Rijn |

# Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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 it is used to identify the sediment transport equation for a specific channel segment or reach (comment 5). This value supersedes 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). |
| ISEDISPLAY | **i** | **1 - NNOD** | Grid element (channel or floodplain) for which the sediment transport capacity for all the sediment transport equations will be listed by output interval TOUT in the SEDTRAN.OUT file. Note that only one equation is used in the actual sediment routing calculations, but the results of all equations are presented 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.<br><br>ISEDSIZEFRAC = 0, No sediment routing by size fraction. Sediment routing is based on the median bed material size $D_{50}$. For this case, the default bed thickness is 10 ft (3m) (see comment 1). |

# Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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 for the sediment size fraction group. |
| JDEBNOD | **i** | **1 - NNOD** | Grid element with the debris basin. |
| SCOURDEP | **i** | **0 - 100**<br>**0 - 30** | Maximum allowable scour depth (ft or m) for all floodplain elements. |
| SEDCHAR | **c** | **M**<br>**C**<br>**Z**<br>**P**<br>**D**<br>**E**<br>**R**<br>**S**<br>**N**<br>**G** | Character line identifier:<br>SEDCHAR = 'M' - Mudflow parameters in Line 1.<br>SEDCHAR = 'C' - Sediment routing parameters in Line 2.<br>SEDCHAR = 'Z' - Sediment routing by size fraction control parameters in Line 3.<br>SEDCHAR = 'P' - Sediment routing by size fraction sediment distribution variables in Line 4.<br>SEDCHAR = 'D' - Debris basin parameters in Line 5.<br>SEDCHAR = 'E' - Sediment scour limitation parameter in Line 6.<br>SEDCHAR = 'R' - Rigid bed grid elements in Line 7.<br>SEDCHAR = 'S' - Sediment supply rating curves in Line 8.<br>SEDCHAR = 'N' - Sediment supply rating curve size fraction distribution in Line 9.<br>SEDCHAR = 'G' - Sediment group.<br>Variable is case sensitive and it must be upper case. |
| 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 bed material sediment that is finer than the representative size diameter. For example, SEDPERCENT = 0.456 defines that 45.6% 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). |

# Variable Descriptions for the SED.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| SGRAD | **r** | **1.0 - 10.** | Sediment gradation coefficient (non-dimensional) for the sediment transport routine. |
| SGSM | **r** | **2.5 - 2.8** | Mudflow sediment specific gravity. |
| SGST | **r** | **2.6 - 2.8** | Sediment specific gravity. |
| 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). SSEDPERCENT represents the percent of the sediment that is finer than the representative size diameter. See SEDPERCENT parameter above. |
| 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:<br>FPN < 0.01 XKX = 24<br>0.01 < FPN < 0.25 XKX = 1,460,865.81* $(FPN)^{2.381}$<br>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. Armouring 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). 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 uncertain as to which equation may be best suited to the 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 FLO-2D Reference Manual to determine an appropriate viscosity and yield stress relationship as function of sediment concentration. There are also mudflow guidelines available in the Handout documents.

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* |
| | 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* |
| W 4 5019.5 27.0 10 1 2 0.5 | |
| 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 element.
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** | **0**<br>**.01 - 1000**<br>**0.25 - 300** | The rate of vertical levee failure (ft/hr or m/hr). Set failrate = 0 for wall collapse. |
| FAILTIME(LF,LD) | **r** | **-99.0 to SIMUL**<br><br>**-99.0** | The duration (hr) that the levee will fail after the FAILEVEL elevation is exceeded by the flow depth.<br>If FAILTIME = 0.0 if the level erosion begins immediately when pipe elevation is exceeded.<br>If FAILTIME > 0.0, the start time for time to 1 ft and time to 2 ft is based on the model start time 0.0 hr.<br>If FAILTIME < 0.0, the start time for time to 1 ft and time to 2 ft is the first dam or levee breach time for multiple breaches.<br>If FAILTIME = -99.0, the start time for time to 1 ft and time to 2 is the first dam or levee breach time for multiple breaches and FAILTIME is reset to 0.0 hr.s (see comment 9). |
| FAILWIDRATE (LF,LD) | **r** | **0**<br>**.01 - 1000**<br>**0.25 - 300** | The rate at which the levee breach widens (ft/hr or m/hr). Set failwidrate = 0 for wall collapse. |
| 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. Variable is case sensitive and it must be upper case. |
| 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. |

# Variable Descriptions for the LEVEE.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| ILEVFAIL | **s** | **0, 1 or 2** | Switch to identify levee failure mode (see comment 6).<br><br>ILEVFAIL = 0 no levee failure.<br>ILEVFAIL = 1 prescribed level failure rates of breach opening or wall collapse.<br>ILEVFAIL = 2 initiate 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:<br><br>1 = north 5 = northeast<br>2 = east 6 = southeast<br>3 = south 7 = southwest<br>4 = west 8 = northwest |
| LEVBASE(LF,LD) | **r** | **0 or**<br>**0 - ∞** | The prescribed final failure elevation. Vertical failure growth stops at this elevation.<br><br>Set LEVBASE = 0 if the levee fails to the floodplain elevation. |
| LEVCHAR(L) | **c** | **L, D, F, W, C or P** | Character Identifier for Lines 2 - 7<br><br>'L' = Line 2<br>'D' = Line 3<br>'F' = Line 4<br>'W' = Line 5<br>'C' = Line 6<br>'P' = Line 7<br><br>Variable is case sensitive and it must be upper case. |
| LEVCREST(L,IM) | **r** | **.01 - 30,000**<br>**.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).<br><br>1 = north 5 = northeast<br>2 = east 6 = southeast<br>3 = south 7 = southwest<br>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 |
|---|---|---|---|
| LEVFRAGRID(LP) | **i** | **1 - NNOD** | Individual levee grid element with fragility curve assignment. The fragility curves must be assigned in BREACH.DAT. |
| LFAILGRID(LF) | **i** | **1 - NNOD**<br>**or**<br>**-1 to -NNOD** | The floodplain grid element number with a levee that may potentially fail.<br>LFAILGRID = 1 to NNOD; Prescribed failure starts at LFAILGRID.<br>LFAILGRID = -1 to -NNOD; Prescribed failure is globally assigned to all levee elements (see comment 1). |
| LGRIDNO(L) | **i** | **1 - NNOD**<br>**or**<br>**-1 to -NNOD** | The grid element number containing the levee segment.<br>LGRIDNO = 1 to NNOD; default no overtop flows reported.<br>LGRIDNO = -1 to -NNOD; overtop flow rates reported to OVERTOP.OUT (see comment 8). |
| RAISELEV | **r** | **0 - 100**<br>**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 prescribed levee failure criteria are as follows:
   a. For the levee to fail when overtopped by the flow, set FAILELEV and FAILTIME = 0.
   b. To fail the levee at a specified elevation, set FAILELEV equal to the failure elevation.
   c. To fail the levee at a specified level below the top of the levee, set FAILELEV to a value less than 10 ft and the levee will fail at an elevation equal to LEVCHREST - FAILELEV.
   d. To fail the levee at a specific level below the crest after the water surface reaches FAILEVEL for a cumulative duration, assign FAILTIME.
   e. To fail the levee to a new base elevation that is different than the floodplain elevation, assign LEVBASE.
   f. To fail a levee to a specified maximum width, set the FAILWIDTHMAX to the limiting width.
   g. To simulate instantaneous collapse, set the FAILRATE and FAILWIDRATE to zero (see Comment 10).
   h. 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.
   i. 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 width is 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 greater than the grid element side width, the breach will extend into adjacent grid elements until the maximum failure width is reached or the levee ends.
4. Flow over a levee is computed as broadcrested weir flow using a coefficient of 3.09 until the tailwater depth is 80% of the headwater depth. The discharge computation 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 erosion failure.

7. Guidelines on levee failure can be found in the Handouts folder: C:\ users\public\public documents\FLO-2D PRO Documentation\flo_help\Handouts\FLO-2D Levee, Dam, and Wall Failure Guidelines.pdf.

8. A report of the levee overtop discharge results are written to the LEVEEOVERTOP.OUT file for any element that is listed with a negative grid element number in the LEVEE.DAT file.

9. A distinction has been made for the start times of the Time to 1 ft and Time to 2 ft and Time to Peak for levee and dam breach models. Two start times are now available. The default start time initiates when the model starts. The alternate start time initiates when the levee or dam breach begins. This is complicated if there are multiple levee or dam breaches. It should also be noted that inflow hydrographs or rainfall may not sync with a breach. There can only be one start time. Distinguishing flows mixing from breaches, flood hydrographs, rainfall is impossible.

10. Wall failure procedures are defined in the guidelines listed above. The proceedures for setting up walls and wall failure, wall failure and grid element elevations, walls and ARFs, and the automatic controls aplied by the FLO-2D engine are all explained in the guidelines.

# 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)**

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 1 and 2,<br>'P' = Line 1<br>'X' = Line 2<br>Variable is case sensitive and it must be upper case. |
| 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:<br>1 flow to the north 5 flow to the northeast<br>2 flow to the east 6 flow to the southeast<br>3 flow to the south 7 flow to the southwest<br>4 flow to the west 8 flow to the northwest<br>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:<br>-1 flow from south only -5 flow from southwest only<br>-2 flow from west only -6 flow from northwest only<br>-3 flow from north only -7 flow from northeast only<br>-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 assigned 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:

TABLE 4.3. 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 more than once, the simulation will fail and the ERROR.CHK file will report the redundant element.

3. The floodplain cross section grid elements can be selected graphically with the GDS or QGIS programs. See FLO-2D Plugin User Manual for instructions.

4. Select a flow direction perpendicular to the cross section only. For example, if the cross section orientation is East to West, the flow direction should be North or South only.

# 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 breach 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 dam levee is continuous through adjoining grid elements (default = grid element octagon side). |
| BREACHTIME | **r** | **- SIMULT to SIMULT**<br><br>**-99** | The cumulative duration (hrs) that the levee erosion will initiate after the water surface exceeds the specified pipe elevation BRBOTTOMEL.<br><br>If BREACHTIME = 0 if the level erosion begins immediately when pipe elevation is exceeded.<br><br>If BREACHTIME > or = 0.0, the start time for time to 1 ft and time to 2 ft is based on the model start time 0.0 hr.<br><br>If BREACHTIME < 0.0, the start time for time to 1 ft and time to 2 ft is the first dam or levee breach time for multiple breaches.<br><br>If BREACHTIME = -99.0, the start time for time to 1 ft and time to 2 is the first dam or levee breach time for multiple breaches and BEACHTIME is reset to 0.0 hr. |
| 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**<br>**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**<br>**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. |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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.<br>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. |
| 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). Variable is case sensitive and it must be upper case. |
| 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. |
| 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). |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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 GBRBOT-WIDWAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side). |
| GBREACHTIME | **r** | **0 - ∞** | The cumulative duration (hrs) that the levee erosion will initiate after the water surface exceeds the specified pipe elevation BRBOTTOMEL. GB-REACHTIME = 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. GBRTOPWIDMAX = 0.0 if the levee is continuous through adjoining grid elements (default = grid element octagon side). |
| GCOHC | **r** | **0 - 750**<br>**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**<br>**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. |
| 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 and a core is present, Manning's n-value for the core material will computed from Strickler's equation. This results in a very low n-value and is 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. |
| 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. This is 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. |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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**<br>**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. |
| 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**<br>**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**<br>**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. |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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. |
| 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<br>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<br>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. |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IBR | c | **B1, B2, D1, D2, D3, D4, G1, G2, G3, G4 or F** | Character line identifier:<br>'G1-G4' = global data<br>'B1' = Global data not related to local breach;<br>'B2' = Grid element and direction<br>'D1-D4' = individual prescribed grid element breach data.<br>'F' = fragility curve data<br>Variable is case sensitive and it must be upper case. |
| IBREACHDIR | i | **1 - 8** | Direction of the specified breach failure in a given grid element. The possible flow directions are:<br>1 = north 5 = northeast<br>2 = east 6 = southeast<br>3 = south 7 = southwest<br>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 transport 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 most likely 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 most likely corresponding to a levee piping failure close to the toe of the levee. |

# Variable Descriptions for the BREACH.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| 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. |
| 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<br>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<br>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. |
| 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. There is a choice of either identifying a global or local levee breach location. If the breach location is assigned locally, it is necessary to provide only the Local (D-Lines) 5-8 in the BREACH.DAT file. This data is entered in the Individual tab of the GDS Breach dialog box. If the model locates all the potential breach locations based on the water surface elevation, then it is only necessary to assign the global parameters (G-Lines) in lines 1-4 and the variable GBRBOTTOMEL = vertical distance (ft or m) below the levee or dam crest elevation. If the water surface elevation exceeds GBRBOTTOMEL, then a levee piping failure will be initiated. One or more breach locations can be prescribed and still permit the model to determine any other potential breach locations to be initiated 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 hard coded to 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, then a prescribed breach location will be simulated as defined by the breach element and flow direction in Line B2 (global breach data is not required)

# FILE: FPFROUDE.DAT

## FLOODPLAIN LIMITING FROUDE NUMBERS



### FPFROUDE.DAT File Variables

F 1 0.65 Line 1: **IFR = 'F' I FROUDEFP (I = 1, NNOD)**

### 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 |
|---|---|---|---|
| IDUM | **i** | **1 - NNOD** | Grid element number (I) of the floodplain grid system. |
| IFR | **c** | **F** | Character Line Identifier = 'F'. Variable is case sensitive and it must be upper case. |
| FROUDEFP | **r** | **0.1 - 2** | Floodplain limiting Froude number. |

# Instructional Comments for the FPFROUDE.DAT File

1. The spatially variable limiting Froude number supersedes the global limiting Froude number (FROUDL) in CONT.DAT.

2. When FROUDEFP is exceeded the grid element roughness value will be increased by 0.001 for the next timestep. After a flood simulation, review ROUGH.OUT to determine where FROUDEFP was exceeded and the maximum n-values for that cell were computed. Consider revising the n-values in the MANNINGS_N.DAT file to match those in the ROUGH.OUT file. This will ensure that FROUDEFP is not exceeded. Rename the MANNINGS_N.RGH file to MANNINGS_N.DAT.

# FILE: SWMMFLO.DAT

## STORM DRAIN DATA FILE

### SWMMFLO.DAT File Variables

Line 1: **SWMMCHAR= 'D' SWMM_JT(I), SWMM_IDEN(I), INTYPE(I), SWMMlength(I), SWMMwidth(I), SWMMheight(I), SWMMcoeff(I), FEATURE(I),CURBHEIGHT(I)**

I = number of storm drain inlet nodes.

D 14291 I37CP1WTRADL 2 13.00 1.00 0.42 2.30 0 0.00

### SWMMFLO.DAT File Example

```
D 14291  I37CP1WTRADL   2  13.00  1.00  0.42  2.30  0  0.00
D 14481  I37CP2WTRADL   2  13.00  1.00  0.42  2.30  0  0.00
D 13785  I14CP1WTRCLRL  2  20.00  1.00  0.42  2.30  0  0.00
D 13968  I14CP2WTRCLRL  2  20.00  1.00  0.42  2.30  0  0.00
D 14156  I15WTRCLRL     3  11.00  7.00  0.50  3.00  0  0.00
```

# Variable Descriptions for the SWMMFLO.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| CURBHEIGHT(I) | **r** | **0.0 - ∞** | Curb height used to calculate discharge on inlets for all INTYPE inlets. |
| FEATURE(I) | **i** | **0, 1, or 2** | Switch positions for FEATURE<br>0 = Horizontal Inlet<br>1 = Vertical Inlet<br>2 = Flapgate (see comment 2). |
| INTYPE(I) | **i** | **1 - 5** | Type of storm drain inlets (see comment 1). |
| SWMMCHAR | **c** | **D** | Character line identifier for the SWMM model inlets. Variable is case sensitive and it must be upper case. |
| SWMMcoeff(I) | **r** | **2.50 - 3.50** | Storm drain inlet weir discharge coefficients (see comment 2). |
| SWMMheight(I) | **r** | **0.0 - 2.0** | Type 1 and type 2 gutter inlets storm drain curb opening heights (typically less than 1 ft).<br>Type 3 grate inlets, SWMMheight = grate sag height.<br>Type 5 manhole inlets, SWMMheight = surcharge depth. |
| SWMM_JT(I) | **i** | **1 - NNOD** | Grid elements that contains storm drain inlets or manholes. |
| SWMMlength(I) | **r** | **0.01 - ∞** | Type 1 and 2 - Storm drain inlet curb opening lengths along the curb.<br>Type 3 and 5 grate (gutter) inlets, SWMMlength = grate wetter perimeter or manhole wetted perimeter. |
| SWMM_IDEN(I) | **c** | **Alpha Numeric** | Storm drain inlet name. Inlet name in the SWMM.inp file. Variable is not case sensitive. No spaces in data.<br>Start the name with an i or I to indicate an inlet: im or IM for manholes. This is mandatory. (See comment 1) |
| SWMMwidth(I) | **r** | **0 - ∞** | Type 2 storm drain inlet curb opening sag width.<br>Type 3 grate (gutter) inlets, SWMMwidth = grate opening area.<br>Type 5 manhole area. |

# Instructional Comments for the SWMMFLO.DAT File

1. The Storm Drain Guidelines manual offers a comprehensive overview of storm drain modeling using FLO-2D. The storm drain feature names **must** begin with an i or I to indicate an inlet, im or IM to indicate a manhole. This naming convention will allow the GDS or QGIS and FLOPRO.EXE to identify features that will connect to the surface for flow exchange. The storm drain naming feature need not be overly complicated because locating the features is simple with the various attribute query tools available in QGIS. A simple convention is to name inlets, junctions, outfalls and conduits so that they are easily sorted. For example. i or I for inlets, im or IM for manholes, j or J for junctions, o or O for outfalls.
2. The SWMMFLO.DAT file is automatically generated by GDS or QGIS using the SWMM.inp file data to locate the inlet position and transfer it to the FLO-2D grid system. For an extensive discussion and guidelines, refer to the Storm Drain Manual. 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 or grate with gutter inlet (at grade or depressed - Type 3)
    4) Non-typical inlet e.g. headwall (Type 4)
    5) Manhole with cover (Type 5)

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 (or grate with 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 in the SWMMFLORT data file.

**Type 5** - Manhole.
Weir coefficient: 2.85 - 3.20
Manhole perimeter
Manhole flow area
Surcharge depth

Note: Orifice flow coefficient = 0.67 (hardwired) for all cases.

# FILE: SWMMFLORT.DAT

## STORM DRAIN TYPE 4 RATING TABLE FILE

### SWMMFLORT.DAT File Variables

Line 1: **SWMMCHARRT='D' SWMM_JT(I)**

D 14292

Line 2: **SWMMCHAR = 'N' DEPTHSWMMRT(J,K) QSWMMRT(J,K)**

N 0.0 0.0

### SWMMFLORT.DAT File Example

```
D   254765
N   0.00   0.00
N   0.10   0.00
N   1.46   10..00
N   2.11   20.00
N   2.72   3.00
N   3.44  40.00
N   4.35   50.00
N   5.48  60.00
N   6..79   70.00
N   8.23   90.00
N  11.85   100.00
D   287346
N   0.00   0.00
N   0.10  0.50
...
```

# Variable Descriptions for the SWMMFLORT.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| DEPTHSWMMRT (J,K) | **r** | **0 -** ∞ | Flow depths for the discharge rating table pairs. |
| QSWMMRT(J,K) | **r** | **0 -** ∞ | Discharge values for the storm drain inlet rating table. |
| SWMMCHAR | **c** | **N** | Character line identifier for the rating table new line data. Variable is case sensitive and it must be upper case. |
| SWMMCHARRT | **c** | **D** | Character line identifier for rating table nodes. Variable is case sensitive and it must be upper case. |
| SWMM_JT(I) | **i** | **1 - NNOD** | Grid elements with storm drain inlets. |

# Instructional Comments for the SWMMFLORT.DAT File

1. The SWMMFLORT.DAT file lists the rating table data for Type 4 inlets.
2. For an extensive storm drain component discussion and guidelines, refer to the Storm Drain Manual.

# FILE: SWMMOUTF.DAT

## STORM DRAIN OUTFALL ID DATA FILE

### SWMMOUTF.DAT File Variables

Line 1: **OUTF_NAME(JT) OUTF_GRID(JT) OUTF_FLO2DVOL(JT)**

OUTFALL1 14292 1

### SWMMOUTF.DAT File Example

OUTFALL1 14292 1

...

# Variable Descriptions for the SWMMOUTF.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| OUTF_NAME(JT) | **c** | **Alpha Numeric** | Name of the feature. Variable is not case sensitive. No spaces in the name. (see comment 1). |
| OUTF_GRID(JT) | **i** | **1 - NNOD** | Grid element corresponding to the location of the outfall. |
| OUTF_ FLO2DVOL(JT) | **s** | **0 or 1** | Outfall discharge switch (see comments 2 and 3):<br>0 = off all discharge removed from storm drain system.<br>1 = on allows discharge to be returned to the FLO-2D system. |

# Instructional Comments for the SWMMOUTF.DAT File

1. The list of outfall names and position should correspond to the SWMM.inp file. Do not add spaces to the name.
2. If the discharge cannot physically return to the surface, set the OUTF_FLO2D-VOL to 0.
3. If the flow in the storm drain system can return to the surface, set the switch to the on position = 1.

# FILE: SDCLOGGING.DAT

## STORM DRAIN BLOCKAGE METHOD FILE

### SDCLOGGING.DAT File Variables

**Line 1: SWMMCHAR= 'D' SWMM_JT(I) SWMM_IDEN(I) SWMM_CLOGFAC(I) CLOGTIME(I)**

D 2694 I1 25 0.50

### SDCLOGGING.DAT File Example

```
D    2694    I1    25   0.50
D    3658    I1    25   0.50
D     224    I1    25   0.50
D    5286    I1    25   0.50
D   10257    I1    25   0.50
...
```

# Variable Descriptions for the SDCLOGGING.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| CLOGTIME(I) | **r** | **0 - ∞** | Time to initiate clogging specified by the user. See comment 2. |
| SWMM_ CLOGFAC(I) | **r** | **0 - 100.** | Clogging factor for each inlet node. The value is a percentage (see comment 1). |
| SWMM_IDEN(I) | **i** | **Alpha Numeric** | Grid element corresponding to the location of the inlet. |
| SWMM_JT(I) | **i** | **1 - NNOD** | Grid element with storm clogged storm drain inlets. |
| SWMMCHAR= 'D' | **c** | **D** | Character line identifier for the SWMM model inlets. Variable is case sensitive. |

# Instructional Comments for the SDCLOGGING.DAT File

1. The percent clogging is based on the available flow area of a storm drain inlet. The metal portion of the inlet grate is not included.
2. The time to initiate clogging is based on the starting time of the model not the time of inundation of the storm drain inlet.

# FILE: TOLSPATIAL.DAT

## SPATIALLY VARIABLE TOLLERANCE VALUES



### TOLSPATIAL.DAT File Variables

4554 0.12 Line 1: **IDUM (I) TOL**

### TOLSPATIAL.DAT File Example

```
4554   0.5
4556   0.5
4557   0.5
4889   0.5
...
```

# Variable Descriptions for the TOLSPATIAL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IDUM(I) | **i** | **1 - NNOD** | Nodes that have a spatially variable TOL value. |
| TOL | **r** | **0.001 - 5.** | Spatially variable TOL value that can range from 0.001 ft to 5 ft(0.0003 m to 1.52 m) |

# Instructional Comments for the TOLSPATIAL.DAT File

1. The TOLSPATIAL.DAT file can be used to create spatially variable depression storage. The TOL value prescribes the flow depth for a floodplain or channel grid element below which no flood routing will be performed. It can be assigned spatially variable for the entire grid system. TOL is analogous to a depression storage rainfall abstraction but also can be applied to study Low Impact Development (LID) retention storage. See the LID Reference white paper and Reference Manual section.

# FILE: WSURF.DAT

## WATER SURFACE ELEVATION COMPARISON

### WSURF.DAT File Variables

| | |
|---|---|
| 10 | Line 2: **NWSGRIDS** |
| 4025 200.25 | Line 2: **IGRIDXSEC(M) WSELEV(M)** |

### WSURF.DAT File Example

```
10
 139   4793.00
 1521   4786.00
 4099   4775.00
 5713   4767.00
 7611   4760.00
 9183   4752.00
10751   4745.00
12442   4736.00
14079   4730.00
15977   4722.00
18061   4711.00
```

# Variable Descriptions for the WSURF.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IGRIDXSEC(M) | **i** | **1 - NNOD** | Nodes that have a known water surface elevation. |
| NWSGRIDS | **i** | **1 - NNOD** | Number of rows in the table of water surface elevations. |
| WSELEV | **r** | **0 -** $\infty$ | Water surface elevation at a given time. |

# Instructional Comments for the WSURF.DAT File

1. The WSURF.DAT file is used as a calibration tool. It is set up with a known peak water surface elevation. When the model completes, a comparison file is written. It compares the high water mark to the max water surface elevation at the corresponding node. The engine will read this file if it is present. There is no need to turn on a switch.
2. The GDS does not build this file. It is created by the user in a text editor program.

# FILE: WSTIME.DAT

## WATER SURFACE ELEVATION COMPARISON

### WSTIME.DAT File Variables

| | |
|---|---|
| 10 | Line 2: **NWSGRIDS** |
| 4025 200.25 12.5 | Line 2: **IGRIDXSEC(M) WSELEVTIME(M) WSTIME(M)** |

### WSTIME.DAT File Example

```
10
 139   4793.00   25
 1521   4786.00   25
 4099   4775.00   25
 5713   4767.00   25
 7611   4760.00   25
 9183   4752.00   25
10751   4745.00   25
12442   4736.00   25
14079   4730.00   25
15977   4722.00   25
18061   4711.00   25
```

# Variable Descriptions for the WSTIME.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IGRIDXSEC(M) | **i** | **1 - NNOD** | Nodes that have a known water surface elevation. |
| NWSGRIDS | **i** | **1 - NNOD** | Number of rows in the table. |
| WSELEVTIME | **r** | **0 -** $\infty$ | Water surface elevation at a given time. |
| WSTIME | **r** | **0 -** $\infty$ | Time of known watersurface elevation. |

# Instructional Comments for the WSTIME.DAT File

1. The WSTIME.DAT file is used as a calibration tool. It is set up with a known water surface elevation and peak discharge time. When the model completes, a comparison file is populated. It compares the high water mark to the maximum water surface elevation at the corresponding node for a given time. The FLO-2D model will read this file if it is present.
2. The GDS does not build this file. It is created by the user in a text editor program.

# FILE: TIMDEPCELL.DAT

## ARRAY OF GRID ELEMENTS FOR TIME OUTPUT

### TIMDEPCELL.DAT File Variables

1521 Line 1: **IGRID(I)**

### TIMDEPCELL.DAT File Example

```
1521
4099
5713
7611
9183
10751
12442
14079
15977
18061
```

# Variable Descriptions for the TIMDEPCELL.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IGRID(I) | **i** | **1 - NNOD** | Nodes selected to generate the variable time output. |

# Instructional Comments for the TIMDEPCELL.DAT File

1. A time series of specific grid cell hydraulics can be created by generating the TIMDEPCELL.DAT file with a list of the grid cells. Change the ITIMTEP to 5 in the CONT.DAT file. Run the model to generate the TIMDEPCELL.OUT file. This output file contains the temporal hydraulic results for each grid cell specified in the TIMDEPCELL.DAT file.
2. The GDS does not build this file. It is created by the user in a text editor.

# FILE: SHALLOWN_SPATIAL.DAT

## ARRAY OF GRID ELEMENTS FOR SPATIALLY VARIABLE SHALLOW N

### SHALLOWN_SPATIAL.DAT File Variables

1521 0.100 Line 1: **IGRID(I) SHALLOWN(I)**

### SHALLOWN_SPATIAL.DAT File Example

```
1521    0.200
4099    0.150
5713    0.220
7611    0.250
9183    0.190
```

# Variable Descriptions for the SHALLOWN_SPATIAL File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IGRID(I) | **i** | **1 - NNOD** | Nodes selected to assign spatially variable shallow n. |
| SHALLOWN(I) | **r** | **0.01 - 0.99** | Spatially variable shallow n-value (see comment 1) |

# Instructional Comments for the SHALLOWN_SPATIAL File

1. 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:

$$n_d = n_b * 1.5 * e^{-(0.4 \, depth/dmax)}$$

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 elements (assumed to be 3 ft or 1 m). This equation is applied by the model as a default and the user can turn '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 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 about 1.31 times the assigned roughness for a flow depth of 3 ft. Assigning a ROUGHADJ value may reduce unexpected high Froude numbers.

The following rules apply:

If the

| | |
|---|---|
| 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) | $n = n_b * 1.5 * e^{-(0.4 \, depth/dmax)}$ |
| 3 ft (1 m) < flow depth | n = n-value in MANNINGS_N.DAT |

# FILE: GUTTER.DAT
## FLOODPLAIN STREET ELEMENT GUTTER DATA



### GUTTER.DAT File Variables

```
2  0.67   0.020
G   4525   25.0   0.67   0.025   8
```

Line 1: **STRWIDTH CURBHEIGHT STREET_n-VALUE**
Line 2: **GUTTERCHAR IGRID(I) WIDSTR(I) CURBHT(I) XNSTR(I) ICURBDIR(J=1-8)**

I = number of grid elements with gutters
J = curbside flow direction

Notes:
Repeat line 2 for each assigned gutter element.

### GUTTER.DAT File Example

```
20.0   0.67   0.020
20   20.   0.67     0.025     1
27   20.   0.67     0.030     1
28   20.   0.67     0.025     1
29   20.   0.67     0.020     1
30   20.   0.67     0.020     1
50   10.   0.67     0.025     5
```

# Variable Descriptions for the GUTTER File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| CURBHEIGHT | **r** | **0 – 1** | Global assignment of the curb height to all the gutter elements (ft or m). See comment 6. |
| CURBHT(K) | **r** | **0 – 1** | Individual gutter element curb height that supersedes CURBHEIGHT (ft or m). |
| GUTTERCHAR | **c** | **G** | Character line identifier for the gutter new line data.Variable is case sensitive and it must be upper case. |
| ICURBDIR | **i** | **1 - 8** | The side of the gutter element that the curb is located. This is one of the eight flow directions (see comment 2). |
| IGRID(I) | **i** | **1 - NNOD** | Floodplain grid element number (see comment 1). |
| STRWIDTH | **r** | **0 – 100** | Global assignment of the street width to all gutter elements (ft or m). |
| STREET_n-VALUE | **r** | **0.01 - 0.5** | Global assignment of the street n-value to all gutter elements. See comment 4. |
| WIDSTR(K) | **r** | **0 – 100** | Street width for individual gutter elements. WIDSTR supercedes STRWIDTH (ft or m). See comment 3. |
| XNSTR(K) | **r** | **0.01 - 0.5** | Street n-values for individual gutter elements. Supersedes STREET_n-value. |

# Instructional Comments for the GUTTER File

1. The gutter elements are street elements defined by the floodplain (not street component) where the flow in the street will be based on the gutter height and a hard coded 2% cross slope in the street (triangular flow area). This concentrates the flow against the gutter and results in deeper flow depth and higher velocities for floodwave movement. The discharge in the gutter flow area defined by the GUTTER.DAT file parameters is routed between gutter elements. This gutter routing algorithm is not the storm drain curb height option that only increases the head on the storm drain inlet. For the curb height option, the flow is still routed as a rectangular flow overland flow between street grid elements.

   - The gutter is assign to one of the 8 sides of the gutter element. The following rules govern the flow exchange of gutter street element with the other elements when the flow depth in the gutter element exceeds the tolerance value (TOL):
   - The flow is exchanged with a contiguous gutter element based on the flow depth against the curb.
   - The flow is shared with a street element without a gutter based on the average depth between the two contiguous elements.
   - The flow is shared with a contiguous floodplain element that is not a street and is not a curb flow direction based on the average flow depth between the two contiguous elements.
   - If the flow direction is the curb direction or one of the two diagonal directions associated with the curb direction, the flow is first exchanged to the sidewalk area within the gutter element when the flow depth exceeds the curb height. This exchange occurs in either direction from the street to the sidewalk or from the sidewalk to the street. After the internal flow exchange within the gutter element is complete the overland flow between the sidewalk area and the contiguous floodplain element in the curb direction is exchanged based on the average flow depth between the two grid elements.

2. If the street width (WIDSTR) exceeds the grid element width, then the street width is limited to 0.90 times the grid element width to allow for the sidewalk surface area.

3. The flow exchange between the gutter street elements is based on the Dept. of Transportation gutter flow modification to account for the larger hydraulic radius. This essentially increases the steady uniform n-value for open prismatic flow by about 5.3 times.

4. The spatially variable or individual element assigned street width, curb height and n-value supersede the global values.

# FILE: BUILDING_COLLAPSE.DAT

## BUILDING COLLAPSE PARAMETERS

### BUILDING_COLLAPSE.DAT File Variables

| | |
|---|---|
| 0 | Line 2: **IARFSMASHGLOBAL** |
| 4025 1 | Line 2: **IG(M) IARFSMASH(M)** |
| | **M = number of grid elements to be considered for collapse** |
| | **IG = grid element** |

### BUILDING_COLLAPSE.DAT File Example

```
0
4563   2
6756   1
23145  1
23146  2
23147  3
25331  4
26345  1
…
```

# Variable Descriptions for the BUILDING_COLLAPSE.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IG(M) | **i** | **1 - NNOD** | Individual grid elements with building that are to be assigned a vulnerability curve for potential collapse. |
| IARFSMASHGLOBAL | **i** | **1 - 4** | Building global vulnerability curve (see Comment 1).<br>1 = Poor<br>2 = Moderate<br>3 = Good<br>4 = Clausen and Clark |
| IARFSMASH | **i** | **1 - 4** | Individual global vulnerability curve (see Comment 1).<br>1 = Poor<br>2 = Moderate<br>3 = Good<br>4 = Clausen and Clark |

# Instructional Comments for the BUILDING_COLLAPSE.DAT File

1. During a flood event or a mud/debris flow, it is possible that a building could collapse and be removed. To predict the collapse of a building during flooding vulnerability curves are applied. The three vulnerability curves are poor or highly susceptible to flood collapse (mobile homes), moderate for buildings with foundations, and good for building with more substantial construction. A fourth curve developed by Clausen and Clark in 1990 is also available. Find more information in the FLO-2D Pro Reference manual.
2. The building collapse routine can also be activated by assigning a negative value to a completely blocked ARF value or to partially blocked ARF values in the ARF.DAT file.

# FILE: OUTRC.DAT

## SURFACE WATER RATING TABLES

### OUTRC.DAT File Variables

| | |
|---|---|
| N 13562 | Line 1: **IVOLSTOCHAR NNODSTOVO** |
| P 1.25 20.5 | Line 2: **IVOLSTOCHAR DEPTHRT(I,K) VOLRT(I,K)** |
| | **I = Depths** |
| | **K = Volume corresponding to I depths.** |

### OUTRC.DAT File Example

N 25146
P 0.00 0.00
P 1.00 5.25
P 2.00 25.2
P 3.00 100.32
P 4.00 180.5
...
P 20.5 736.00
N 14079
P 0.00 0.00
P 1.00 2.50
...

# Variable Descriptions for the OUTRC.DAT File

(s) Switch (i) = Integer variable (r) = Real variable (c) = Character

| VARIABLE | FMT | RANGE | DESCRIPTION |
|---|---|---|---|
| IGRIDXSEC(M) | **c** | **N or P** | N = Node line<br>P = storage rating table pairs. |
| NODDSTOVO | **i** | **1 - NNOD** | Grid element with a storage volume rating table (see comment 1). |
| DEPTHRT | **r** | **0 -** $\infty$ | Increment flow depth for the volumetric rating table above the lowest elevation in the grid element topographic data base. |
| VOLRT | **r** | **0 -** $\infty$ | Volume for each incremental depth. |

# Instructional Comments for the OUTRC.DAT File

1. Grid element storage volume rating tables defines variable volume as a function of flow depth instead of a cell having one uniform elevation. This enables the digital terrain data base within to be represented. The rating table depth is based on the lowest elevation within the grid element. At least 25 DTM points are required to enable the rating table to be established. The storage volume rating table is generated by the GDS. The FLO-2D model will read this file if it is present.

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
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 volume conservation. The second screen option is a color graphics display of the grid element 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 for 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.

Output files are created with both spatial and temporal formats. Output files that are listed in the order of the output intervals are temporal output and output files listed in the order of the grid elements is spatial output. 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 at the end of the BASE.OUT 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 for a simulation either make copies of the output files or copy the data files to a new project folder and run the next simulation from it.**

The user does not have to specify any output file names. It is important to note that each time the model 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 simulation, the user should create another subdirectory.

**Table 5.1. List of *.OUT Files and Unit Numbers**

| File No. | File Name | File No. | File Name |
|---|---|---|---|
| 2 | BASE.OUT | 161 | LEVEEDEFIC.OUT |
| 255 | BREACH.OUT | 160 | LEVOVERTOP.OUT |
| 288 | BUILDING_COLLAPSE.OUT | 1601 | LEVOVERTOPMAX.OUT |
| 263 | CHAN_INTERIOR_NODES.OUT | 1594 | LOW_LEVEE_CREST_ELEVATIONS.OUT |
| 152 | CHANBINARY.OUT | 1574 | ManholePop.OUT |
| 345 | CHANLOSSES.OUT | 106 | MAXQBYDIR.OUT |
| 14 | CHANMAX.OUT | 105 | MAXQHYD.OUT |
| 1596 | CHANNEL_CONVERGENCE.OUT | 107 | MAXQRESOLVED.OUT |
| 140 | CHANSEDSIZE.OUT | 124 | MAXWSELEV.OUT |
| 90 | CHANSTABILITY.OUT | 25 | MULTCHN.OUT |
| 93 | CHANWS.OUT | 205 | MULTSTEEP.OUT |
| 74 | CHNBEDEL.OUT | 82 | OUTNQ.OUT |
| 66 | CHVOLUME.OUT | 87 | OVERBANK.OUT |
| 103 | CONFLUENCE.OUT | 659 | PRESCRIBED_BREACHQ.OUT |
| 154 | CROSSBINARY.OUT | 321 | REVISED_RATING_TABLES.OUT |

**Table 5.1. List of *.OUT Files and Unit Numbers**

| | | | |
|---|---|---|---|
| 54 | CROSSMAX.OUT | 157 | SEDBINARY.OUT |
| 79 | CROSSQ.OUT | 18 | SEDCHAN.OUT |
| 1132 | DAMBREACH_VOLUME.OUT | 77 | SEDCONSERV.OUT |
| 1650 | DEBUG_MM_DD_YYYY.OUT | 17 | SEDFP.OUT |
| 61 | DEPCH.OUT | 88 | SEDTRAN.OUT |
| 261 | DEPCHFINAL.OUT | 104 | SPECENERGY.OUT |
| 60 | DEPFP.OUT | 102 | STATICPRESS.OUT |
| 1598 | DEPRESSED_ELEMENTS.OUT | 23 | STREET.OUT |
| 20 | DEPTH.OUT | 153 | STREETBINARY.OUT |
| 28 | DEPTHDUR.OUT | 75 | STRELEV.OUT |
| 528 | DEPTHDUR2.OUT | 73 | STVEL.OUT |
| 27 | DEPTHTOL.OUT | 72 | STVELDIR.OUT |
| 113 | EVACUATEDCHAN.OUT | 11 | SUMMARY.OUT |
| 1131 | EVACUATEDFP.OUT | 65 | SUPER.OUT |
| 62 | FINALDEP.OUT | 55 | SURFAREA.OUT |
| 83 | FINALDIR.OUT | 1565 | SWMMOUTFIN.OUT |
| 81 | FINALVEL.OUT | 1560 | SWMMQIN.OUT |
| 1597 | FLOODPLAIN_CONVERGENCE.OUT | 21 | TIMDEP.OUT |
| 193 | FLOODWAVETIME.OUT | 78 | TIMDEP_NC4.OUT |
| 45 | FLOODWAY.OUT | 212 | TIMDEPCELL.OUT |
| 111 | FPINFILTRATION.OUT | 19 | TIME.OUT |
| 151 | FPLAINBINARY.OUT | 192 | TIMEONEFT.OUT |
| 1563 | FPRIMELEV.OUT | 191 | TIMETOPEAK.OUT |
| 139 | FPSEDSIZE.OUT | 190 | TIMETWOFT.OUT |
| 202 | GDSFILE.OUT | 262 | VELCHFINAL.OUT |
| 204 | GDSRUNEND.OUT | 63 | VELDIREC.OUT |
| 80 | HYCHAN.OUT | 64 | VELFP.OUT |
| 96 | HYCROSS.OUT | 24 | VELOC.OUT |
| 669 | HYDRAULICSTRUCTURE_RUNTIMEWARNINGS.OUT | 641 | VELRESMAX.OUT |
| 70 | HYDROSTRUCT.OUT | 117 | VELTIMEC.OUT |

**Table 5.1. List of *.OUT Files and Unit Numbers**

| | | | |
|---|---|---|---|
| 123 | HYSTREET.OUT | 118 | VELTIMEFP.OUT |
| 155 | HYSTRUCBINARY.OUT | 128 | VELTIMEST.OUT |
| 101 | IMPACT.OUT | 150 | VOLUMEBINARY.OUT |
| 289 | INFIL_DEPTH.OUT | 181 | WSTIME.OUT |
| 67 | INFILHY.OUT | 94 | XSEC.OUT |
| 92 | INTERGWS.OUT | 91 | XSECAREA.OUT |
| 59 | LEVEE.OUT | 156 | XSECSEDBINARY.OUT |
| 22 | ROUGH.OUT | | |

**Table 5.2. List of *.TMP Files and Unit Numbers**

| File No. | File Name | File No. | File Name |
|---|---|---|---|
| 254 | BREACH.TMP | 159 | LEVOVERTOP.TMP |
| 8 | CHMAX2.TMP | 12 | OUTNQ.TMP |
| 16 | CROSSQ.TMP | 76 | OUTNQ2.TMP |
| 13 | HYCHAN.TMP | 112 | OUTNQMAX.TMP |
| 15 | HYCROSS.TMP | 1566 | SWMMOUTFIN.TMP |
| 71 | HYDROSTRUCT.TMP | 1561 | SWMMQIN.TMP |
| 122 | HYSTREET.TMP | | |

## 5.2 Output File Descriptions and Options

Use NotePad© , NotePad++© 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-inclusive 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 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 output file named SUMMARY.OUT that is more complete.

There is so much output data in the BASE.OUT file that the user is encouraged to avoid generating this file. All of the text output in this file is provided in individual ASCII xyz output files for plotting purposes and the user will probably have little interest in the BASE.OUT format of the floodplain hydraulics for the individual grid elements.

This output file can become large and it takes too long to write to it for models with 500,000 grid elements or more. Set NOPRTFP = 2 and it will not be created.

- If NOPRTFP = 0, all the BASE.OUT floodplain flow data is reported.
- If NOPRTFP = 1, the BASE.OUT floodplain outflow data is not reported.
- If NOPRTFP = 2, the entire file is not created.
- If NOPRTFP = 3, only floodplain outflow data is reported to the BASE.OUT file.

*BINARY FILES*

The following binary backup files are generated when IBACKUP = 1. These files can be 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 erosion breach routine is activated for dams or levees. The output is listed by grid element number singular and tabular results. The initial and peak discharge is reported for each grid element and the time each occurred. The tabular data is reported for the breach discharge as follows:

- Time - simulation time output
- Direction - breach direction 1-8 grid element directions
- Breach Q - total discharge through the breach and the end of the interval (cfs or cms)
- Sediment discharge - total sediment through the breach at the end of the interval (cfs or cms)
- Sediment concentration - concentration of sediment
- Bottom width - breach width at the bottom of the dam or levee at the output interval (ft or m)
- Top width - breach width at the top of the dam or levee at the output interval (ft or m)
- Breach elevation - elevation of the bottom of the breach at the output interval (ft or m)

*BUILDING_COLLAPSE.OUT*

This file lists the grid elements with full or partial ARF values that will be reset to 0.0 during the model run to simulate the collapse and removal of buildings. This occurs because the flood depth and velocity exceed the building collapse criteria. The following tabular data is printed:

- Grid element
- Time
- Velocity - velocity at the time of collapse (fps or mps)
- Depth - depth at the time of collapse (ft or m)
- Minimum collapse depth based on the velocity (ft or m)

*CHAN_INTERIOR_NODES.OUT*

A list of all the grid elements between the channel bank elements representing the interior of the 1-D channel are listed in this file. These elements should reflecting the channel maximum depth when plotting maximum depths in Mapper or GIS applications. The channel bank elements are not included in this file.

*CHANBANKEL.CHK*

This file reports the difference between the channel bank elevation and the grid element elevation for each assigned bank elements. If the bank elevation difference exceeds the specified criteria, the floodplain elevation will be reset to channel bank elevation at runtime. This assumes that the surveyed bank elevation is more accurate than the interpolated floodplain elevation. The bank elevation difference criteria is:

- Channel grid element
- Xcoord
- Ycoord
- Bank elevation (ft or m)
- Floodplain elevation (ft or m)
- Difference (ft or m)

Channel bank elevation is different from the floodplain elevation by 1 ft or more.

If the slope associated with the bank elevation difference based on the grid element side width is greater than 0.01 (1%)

*CHANMAX.OUT*

The maximum discharge and stage for each channel element and the corresponding time of occurrence is written to this file. This file is useful for finding channel cross sections that might be surging. If the timing if the maximum values do not correspond with the peak discharge, the channel element may be surging. The following columns are written:

- Node
- Max Q - Maximum discharge for channel element (cfs or cms)
- Time - Time of Qmax
- Max Stage - Maximum stage for channel element (ft or m)
- Time - Time of max stage

*CHANNEL.CHK*

When the channel cross section width exceeds the grid element width, the cross section needs to extend into 1 or more neighboring elements. When the channel surface area is 0.95 times the floodplain surface area the channel needs to extend into 1 or more neighboring elements. This file lists the necessary extensions.

If a channel right bank is placed on an interior channel element, this file lists the bank that needs to be repositioned.

The file lists any channel / levee conflicts that may need to be fixed.

If the channel cross section is R, T or V (non-natural cross sections) and the channel is extended to more than one grid element and the bank elevations are not assigned in CHAN.DAT. This file lists the difference between the right and left channel bank elevations based on the floodplain elevations in two different bank elements.

### *CHAN.RGH*

CHAN.RGH is a duplicate file of the CHAN.DAT file with the updated Manning's n-value changes that were reported in the ROUGH.OUT file. The maximum and final Manning's n-value changes are listed in the ROUGH.OUT file. To accept the changes to Manning's n-values, CHAN.RGH can be renamed to replace 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.

### *CHANNEL_CONVERGENCE.OUT*

This file lists the channel elements that failed to converge in three passes of the routing algorithm. The solution is then based on the diffusive wave for that element and timestep only. The output files reports:.

- Time - time of failed convergence
- Grid element
- Depth - depth at time of failed convergence (ft or m)
- Velocity - various velocity terms in the solution algorithm (fps or mps)

### *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.

- Grid
- Xcoord
- Ycoord
- Watersurface elevation (ft or m)

*CHNBEDEL.OUT*

The channel grid element number and the final channel bed elevation are presented in this file.

- Grid element
- Elevation - final bed elevation (ft or m)

*CHVOLUME.OUT*

The channel volume distribution is listed in this output file including channel inflow, channel outflow, overbank flow, return flow from the floodplain, infiltration, channel storage and storm drain return flow. Review this file along with the SUMMARY.OUT to determine if the channel flow volume is being conserved.

- Time
- Inflow and rain - (acre ft or cm)
- Channel storage - (acre ft or cm)
- Channel outflow - (acre ft or cm)
- Overbank outflow - (acre ft or cm)
- Return inflow - (acre ft or cm)
- Infiltration - (acre ft or cm)
- Evaporation - (acre ft or cm)
- Outflow to storm drain - (acre ft or cm)
- Inflow from storm drain - (acre ft or cm)
- Volume conservation - (acre ft or cm)

*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 FPX-SEC.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.

- Time
- Discharge - hydrograph for grid element (cfs or cms)

### *DAMBREACH_VOLUME.OUT*

This file reports the prescribed dam breach discharge volume in acre-ft or cubic meters. The data includes total volume for water and sediment.

- Time
- Volume hydrograph (af or cm)

### *DEBUGXX.OUT*

This file reports all data related bugs and conflicts with an error code, grid element and a description of the error, warning or conflict. It is imported by QGIS FLO-2D Plugin so users can visualize data error locations.

### Vo*DEPRESSED_ELEMENTS.OUT*

This file is generated at the end of the data input at runtime. Every grid element elevation is checked with its neighbors' elevations to see if it is depressed below the minimum difference of the DEPRESSDEPTH variable in CONT.DAT and if so, it is listed in this file. A value of DEPRESSDEPTH = 3.0 ft is suggested which will help identify artificial ponded flow conditions. This depth will ignore minor small depression elements which can fill and overview.

- Grid element
- Minimum elevation difference - lowest elevation difference between this element and its neighbors. (ft or m)

### *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.

- Grid
- Xcoord
- Ycoord
- Variable

### *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 except for a hardwired depth of 2 ft.

- Grid
- Xcoord
- Ycoord
- Time

### *ERROR.CHK*

The ERROR.CHK file contains data input error and warning messages and some runtime error messages. The backup data files (*.BAC) can 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. This file is defined in more detail in the troubleshooting section chapter 7.

*EVACUATEDCHAN.OUT*

The channel elements that experience a complete evacuation of the channel volume are listed in this output file. The channel elements in this file should be cross-correlated with those listed in TIME.OUT and VELTIMEC.OUT files.

- Element
- Number of evacuations

*EVACUATEDFP.OUT*

The floodplain elements that experience a complete evacuation of the floodplain volume are listed in this output file. The floodplain elements in this file should be cross-correlated with those preeminently listed in TIME.OUT and VELTIMEFP.OUT files.

- Element
- Number of evacuations

*FLOODPLAIN_CONVERGENCE.OUT*

This file lists the floodplain elements that failed to converge in three passes of the routing algorithm. The solution is then based on the diffusive wave for that element and timestep only. The output files reports:.

- Time - time of failed convergence
- Grid element
- Depth - depth at time of failed convergence (ft or m)
- Velocity - various velocity terms in the solution algorithm (fps or mps)

*FLOODWAVETIME.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 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 2.0 ft or 0.67 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: The time elapsed from the initial failure of the dam until the grid element returns to its pre-flood water elevation (0.1ft) prior to failure. The dam breach initialization is based on the first incremental change in flow depth greater than the tolerance value (TOL).
- 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. See the FLO-2D Reference Manual for a discussion on the floodway routine..

### *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.

- Grid element
- Xcoord
- Ycoord
- Total infiltration (ft or m)

*FPREV.NEW*

This output file reports the differences in elevation between the rim elevation in the SWMM.inp file and the FLO-2D grid element elevation. This file should be reviewed to evaluate the elevations representing the inlet reference elevation.

- Grid element
- New grid element elevation (ft or m)

*FPRIMELEV.OUT*

This output file reports the differences in elevation between the rim elevation in the SWMM.inp file and the FLO-2D grid element elevation. This file should be reviewed to evaluate the elevations representing the inlet reference elevation.

- Grid element
- Floodplain elevation - grid element elevation (ft or m)
- Rim elevation - rim elevation of storm drain inlet or manhole (ft or m)
- Difference (ft or m)
- New floodplain elevation - elevation the model uses (ft or m)

*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.

The file is arranged in tables by grid element.

- Grid element
- Sediment diameter. (mm)
- Percent finer initial
- Percent finer final

*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. The following columns are printed for each channel element.

- Time - output interval
- Elevation - watersurface elevation starting at bed elevation.
- Thalweg depth - average depth above the lowest point in the channel for the duration of the output interval. (ft or m)
- Velocity - depth average velocity for cross section for the duration of the output interval (fps or mps)
- Discharge - average discharge for the output interval (cfs or cms)
- Froude number - based on the average depth and velocity.
- Flow area - average flow area given by the average discharge divided by the average velocity (sqft or sqm)
- Wetted Perimeter - average wetted perimeter for the cross section for the duration of the output interval (ft or m)
- Hydraulic radius average flow area divided the average wetted perimeter (ft or m)
- Top width - average top width for the duration of the output interval (ft or m)
- Width to depth ratio - average width divided by the average depth
- Energy slope - average water surface head plus the average velocity head divided by the length of the channel between grid element centers
- Bed shear stress - average energy slope times the average hydraulic radius times gamma (specific weight of water)
- Surface area - average surface area of the channel (top width times channel length) for the duration of the output interval (sqft or sqm)

### *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..

- Time
- Top width - distance between the first and last node (ft or m)
- Depth - average depth across the complete cross section (ft or m)
- Velocity - average velocity for the complete cross section (fps or mps)
- Discharge - resolved and compiled discharge for the complete cross section. This is the most important value (cfs or cms)

### *HYDROALL.OUT*

This file is generated by the HYDROG.EXE. It is used internally and not by the end user.

### *HYDROSTRUCT.OUT*

The discharge hydrographs of all the hydraulic structures is presented in this output file. This file lists time and the discharge seen an the inlet and at the outlet for each hydraulic structure. If the values are negative in the inlet, the water is moving from the outlet to the inlet as backwater. If the discharge varies wildly, there could be surging. The rating table or curve might not match the cross sectional areas adjacent to the structures.

- Time
- Discharge inlet
- Discharge outlet

### *HYSTREET.OUT*

The street flow hydrograph for the grid element that is coincidental to the street and the cross section is recorded in this file.

### *IMPACT.OUT*

This file displays the impact force per linear foot of structure on the flood-plain. The data is presented by grid element number, coordinates and force per linear foot or meter.

- Grid element
- Xcoord
- Ycoord
- Impact - force per linear

### *INFILHY.OUT*

The hydraulic conductivities are listed in this file to review their spatial variation. This file contains grid element number, x- and y-coordinates and floodplain hydraulic conductivity.

- Grid element
- Xcoord
- Ycoord
- Hydraulic conductivity

*INFIL_DEPTH.OUT*

- Grid element
- Xcoord
- Ycoord
- Soil depth - assigned limiting infiltration soil depth (ft or m)
- Infiltration depth - total infiltration depth (ft or m)
- Stop - 0 or 1, where 1 = available infiltration depth was filled and infiltration stopped

*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.

- Grid element
- Water surface elevation (ft or m)

*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.

- Grid element
- Direction - fail direction 1-8
- Time overtopped

*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.

- Grid element
- Xcoord
- Ycoord
- Levee deficit

### *LEVOVERTOP.OUT*

The discharge hydrograph overtopping the levee within the grid element is reported in this file. Only those levee grid elements with a negative levee element number in LEVEE.DAT will be reported when overtopped. The discharge is combined for all the potential levee overtopping directions for the grid element.

- Grid element
- Discharge total
- Time - time of overtopping,
- Discharge direction negative value means flow is moving from the opposite grid to the grid with the levee assigned

### *LEVOVERTOP.OUT*

The max discharge of the water overtopping the levee within the grid element is reported in this file. Only those levee grid elements with a negative levee element number in LEVEE.DAT will be reported when overtopped. The discharge is combined for all the potential levee overtopping directions for the grid element.

- Grid element
- Discharge max (cfs or cms)
- Time - time of overtopping (hrs)

### *LOW_LEVEE_CREST_ELEVATIONS.OUT*

Levee crest elevations that are less than a minimum difference above the ground are list in this file. The minimum elevation difference is the DEPRESSDEPTH parameter in the CONT.DAT file. This variable is used to evaluate the minimum difference in the levee crest elevations compared to the ground elevation on both sides of the levee. If used with DEPRESSED_ELEMENTS.OUT, the DEPRESSDEPTH variable either has to be the same value or two separate independent simulations are required for different values (use SIMUL = 0.1 or 0.01 hrs for each).

- Grid element - element with the levee assigned
- Neighbor grid element - element across from the levee cutoff direction
- Direction - levee cutoff direction 1-8
- Levee crest elevation (ft or m)
- Ground elevation (ft or m)
- Elevation difference (ft or m)

### *MAXQBYDIR.OUT*

This output file lists the maximum floodplain grid element discharge according to the eight flow directions and the time of occurrence.

- Grid element
- North - Qmax (cfs or cms) Time
- NE - Qmax (cfs or cms) Time
- East - Qmax (cfs or cms) Time
- SE - Qmax (cfs or cms) Time
- South - Qmax (cfs or cms) Time
- SW - Qmax (cfs or cms) Time
- West - Qmax (cfs or cms) Time
- NW - Qmax (cfs or cms) Time

*MAXQHYD.OUT*

This output file lists the maximum floodplain grid element discharge and the associated hydraulics including:

- Grid element
- Time
- Maximum discharge (cfs or cms)
- Direction - direction max discharge was recorded 1-8
- Water surface
- Depth (ft or m)
- Velocity (fps or mps)
- Combined Qmax (cfs or cms)
- Direction - direction max velocity 1-8

*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.

- Grid element
- North - Qmax (cfs or cms)
- NE - Qmax (cfs or cms)
- East - Qmax (cfs or cms)
- SE - Qmax (cfs or cms)
- South - Qmax (cfs or cms)
- SW - Qmax (cfs or cms)
- West - Qmax (cfs or cms)
- NW - Qmax (cfs or cms)

*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.

- Grid element
- Xcoord
- Ycoord
- Water surface elevation (ft or m)

### *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:

- Grid element
- Max width (ft or m)
- Depth (ft or m)
- Qmax (cfs or cms)

### *TIMDEP_NC4.OUT*

The TIMDEP_NC4.OUT file is generated when the time series output in the control data file is set to 3 or 4. The file has a complete dataset for each cell written at the output interval.

- J= cell number
- FLOWTIMDEP= water depth for each cell
- QRESMAX= max peak discharge of the 8 directions in the cell
- QRESMAXDIR= direction for the max peak discharge in the cell
- MAXVEL= magnitude of the maximum velocity of the 8-directions
- MAXVELLOC= direction for the maximum velocity (8 directions)
- CUMIN_CUMOUT(J)= cumulative flow (IN-OUT) in the cell
- SUREXCH = surface exchange index (equal to 1 is at least one of the 8 directions has inflow or outflow)
- WSEL= Water Surface Elevation for each cell.

*MULTSTEEP.OUT*

This file lists the number of steep multiple channels found within the assigned minimum and maximum slopes.

*OUTNQ.OUT*

The OUTNQ.OUT file is separated into two data areas. The first section contains a summary of the maximum discharge, time of peak and the discharge hydrograph for each floodplain outflow element. The second section is column data that includes the following for each outflow node:

- Grid element
- Time
- Discharge (cfs or cms)

*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.

- Grid element
- Xcoord
- Ycoord
- Time
- Water surface elevation - elevation at time water goes overbank (ft or m)
- Thalweg depth - depth at time water goes overbank (ft or m)
- Velocity - average velocity at time water goes overbank (fps or mps)
- Discharge - q at time water goes overbank (cfs or cms)
- Overbank volume
- Available floodplain area

*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 or because the rating table has insufficient low depth stage-discharge pairs or the cross section do not match the rating table data.

### *ROUGH.OUT*

The ROUGH.OUT file reports the automated Manning's n-value adjustment during model simulation including n-value change for exceeding the Courant number and exceeding the limiting Froude. The user specifies a maximum Froude number for overland, channel and street ?ow. When the computed Froude number exceeds the defined maximum value for a given grid element, the n-value for that grid element is increased by a value based on the percent change in the n-value. During the falling limb of the hydrograph when the Froude number is no longer exceeded, the n-value is decreased by 0.0005 until the original n-value is reached. When the Courant number timestep is exceeded consecutive times by the same grid element, then n-value is also increased. With increasing consecutive timestep decrements, the increase in n-value decreases. The maximum n-value, time of occurrence, and original n-values for floodplain, channel and street are listed in ROUGH.OUT by grid element.

### *SD MANHOLEPOPUP.OUT*

SDManholePopUp.OUT is created when at least one manhole pops in the storm drain system. This file contains the following information:

- Xcoord
- Ycoord
- Grid element
- Manhole ID
- Time
- Pressure Head
- Rim elevation + Surcharge Elevation
- FLO-2D WSE.

### *SEDCHAN.OUT*

The sediment transport routine will compute scour and deposition in the channel.

- Grid element
- Xcoord
- Ycoord
- Maximum deposition (ft or m)
- Maximum scour (ft or m)
- Final bed elevation difference (ft or m)
- Maximum water surface elevtion (ft or m)

### *SEDCONSERV.OUT*

The sediment transport conservation summary is listed by output interval.

- Time
- Inflow (cuft or cum)
- Floodplain storage (cuft or cum)
- Channel storage (cuft or cum)
- Street storage (cuft or cum)
- Outflow (cuft or cum)
- Conservation total (cuft or cum)
- Conservation percent (cuft or cum)

### *SEDFP.OUT*

Similar to the SEDCHAN.OUT file, the floodplain scour and deposition are reported in the SEDFP.OUT file.

- Grid element
- Xcoord
- Ycoord
- Maximum deposition (ft or m)
- Maximum scour (ft or m)
- Final bed elevation difference (ft or m)
- Maximum water surface elevation (ft or m)

### *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 for a single specified grid element. Set the variable to print the file in the SED.DAT file or the GDS.

- Zeller/Fullerton
- Yang
- Englund/Hansen
- Ackers/White
- Laursen
- Toffaleti
- MPM-Woo
- MPM-Smart
- Karim/Kennedy
- Parker/Klingemen/McClean
- Van Rijn

## *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.

- Grid element
- Xcoord
- Ycoord
- Specific energy (ft or m)

## *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.

- Grid element
- Xcoord
- Ycoord
- Static pressure (lb/ft or N/m)

## *STORMDRAIN_ERROR.CHK*

Storm drain error and warning messages are written to this file. The error/warnings related to conflicts between storm drain features and surface components as well as the elevations checks are listed. The Storm Drain Guidelines manual has a troubleshooting section that will help determine how the errors and conflicts can be corrected.

## *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 street elements that exceeded 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.

- Grid element
- Xcoord
- Ycoord
- Maximum street depth (ft or m)

*STRELEV.OUT*

Final street elevations used in the model simulation are listed in this file.

- Grid element
- Final street elevation (ft or m)

*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.

Mass balance information for the various flow components is reported.

- Inflows
    - Inflow hydrograph volume
    - Rainfall volume
- Storage
    - Floodplain storage
    - Channel storage
    - TOL storage (see TOLER.DAT)
- Outflow
    - Infiltration and interception
    - Floodplain outflow
    - Channel infiltration

Storm drain exchange volume is reported

- Storm drain inflow
- Total inflow
- Total outflow
- Storm drain return flow
- Storm drain mass balance

Storm drain volume data from swmm.rpt

- Wet weather inflow
- External inflow
- External outflow
- Return flow to surface
- Total storm drain storage
- Continuity error

Totals are reported

- Total outflow
- Total volume and storage
- Area of inundation data

- Wetted floodplain area
- Wetted channel area

Project Specific Data

- Grid element size
- Total number of grid elements
- Grid System area (acres or m^2 and mi^2 or km^2)

Average hydraulics

- Discharge (cfs or cms)
- Velocity (fps or cms)
- Flow area (ft^2 or m^2)
- Flow depth (ft or m)
- Flow width (ft or m)

Computation data

- Total Computations
- Computer run time (hrs)
- Termination date and time

### *SUPER.OUT*

Instead of writing the supercritical flow messages at runtime (and limiting them to the first 100 or so instances), the maximum supercritical Froude number (associated depth and time and number of occurrences) are tracked and sorted by Froude number in descending order at model termination for both floodplain and channel (at the bottom of the file). It also indicates if the grid elements are hydraulic structures. By correlating this file with TIME.OUT, ROUGH.OUT, VELTIMEFP.OUT, the user can address the problematic elements with greater insight.

- Grid element
- Max Froude number
- Depth (ft or m)
- Time
- Number of supercritical timesteps

### *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.

The SURFAREA.OUT file enables a review of the surface area distribution between the various components.

- Grid element
- Arf-reduced area - total area minus the building
- Channel area - bank elements covered by part of the channel
- Street area - area covered by street component
- Mult channel area - area covered by mult channel
- Overland area - remaining area not covered by a component
- Mult channels - switch tells the user this element has a mult channel.

### *SWMM.OUT*

This is the output binary file that contains the numerical results from a storm drain simulation. This file is used by storm drain interface (GUI) to create the time series plots and tables, profile plots, and statistical analyses. The FLO-2D Storm Drain Guidelines manual is the best resource for developing, troubleshooting and reviewing anything storm drain related. For more information look at: C:\Users\Public\Documents\FLO-2D PRO Documentation\flo_help\Manuals\FLO-2D Storm Drain Manual.pdf.

### *SWMM.RPT*

This file contains the report information and the results of the storm drain flood routing in ASCII Format. The storm drain model engine generates this file. It is extensive and contains discharge hydrographs for every drain inlet, outlet and conduit. The Storm Drain Guidelines manual is the best resource for developing, troubleshooting and reviewing anything storm drain related. For more information look at: C:\Users\Public\Documents\FLO-2D PRO Documentation\flo_help\Manuals\FLO-2D Storm Drain Manual.pdf.

### *SWMMOUTFIN.OUT*

This file reports the storm drain outfall hydrographs for return flow to the surface water system. This file lists the grid element (or channel element if applicable) followed by the time and discharge pairs. The Storm Drain Guidelines manual is the best resource for developing, troubleshooting and reviewing anything storm drain related. For more information look at: C:\Users\Public\Documents\FLO-2D PRO Documentation\flo_help\Manuals\FLO-2D Storm Drain Manual.pdf.

### *SWMMQIN.OUT*

The discharge hydrograph and return flow (time, discharge and return flow) into each storm drain inlet of the pipe network is reported in this file. Each inlet has a discharge hydrograph and return flow reported each output interval TOUT timestep. The Storm Drain Guidelines manual is the best resource for developing, troubleshooting and reviewing anything storm drain related. For more information look at: C:\Users\Public\Documents\FLO-2D PRO Documentation\flo_help\Manuals\FLO-2D Storm Drain Manual.pdf

### *SD ManholePopUp.OUT*

This file reports the storm drain manhole nodes that have enough pressure head to pop off the manhole cover. The pop off presure head is an instantaneous head that removes the manhole cover. This pressure head can be different to the reported pressure head in the SWMM.RPT file.

- Manhole ID
- Popped time
- Pressure head pop off must be greater than the following:
- Rim and surcharge head
- FLO-2D water surface elevation

### *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 - output interval for time series. Single value at the top of the columns.

- Grid element
- Depth (ft or m)
- Velocity (sqrt(x^2+y^2)) (fps or mps)
- Velocity x - velocity vector x
- Velocity y - velocity vector y
- Water surface elevation (ft or m)

### *TIMDEPCELL.OUT*

This file contains flow depth, velocity, and velocity direction x and y, and water surface elevation for a set of grid elements defined by the TIMEDEPCELL.DAT file. The user specifies time intervals with TIMTEP in CONT.DAT.

*TIMDEP.HDF5*

This binary output 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 output time intervals (TIMTEP in CONT.DAT). This file is written in binary format (HDF5) and it has the same results than the TIMDEP.OUT file.

*TIMDEP_NC4.OUT*

This file contains specific details for every grid element at each time series output interval. The user specifies output time intervals with TIMTEP in CONT.DAT. This is an ASCII file.

- Grid element
- Depth (ft or m)
- Qmax (cfs or cms)
- Qmax direction - grid element direction 1 - 8
- Vmax (fps or mps)
- Vmax direction - grid element direction 1 - 8
- Qnet - all flow in minus all flow out (cfs or cms)
- Surface Exchange - switch 0 or 1 identifies if cell had any flow for the time interval

*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.

- Grid element - floodplain, channel, or street
- Number of timestep decrements
- Percent change in depth
- CFL Stability criteria
- Dynamic wave stability criteria

The file lists the last one hundred time step decreases and the node type.

*TIMEONEFT.OUT*

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. This file is typically used for dam and levee breach analysis.

- Grid element
- Xcoord
- Ycoord
- Time to one ft depth

*TIMETOPEAK.OUT*

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. While this file is typically used for dam and levee breach analysis, it valid for general flood studies.

- Grid element
- Xcoord
- Ycoord
- Time to one ft max depth

*TIMETWOFT.OUT*

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. This file is typically used for dam and levee breach analysis.

- Grid element
- Xcoord
- Ycoord
- Time to two ft depth

*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.

- Grid element
- Xcoord
- Ycoord
- Velocity in the channel element (fps or mps)

*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.
FINALDIR.OUT - Flow maximum velocity direction at the end of the simulation.

*VELRESMAX.OUT*

This file lists the maximum resolved velocities as a vector field. It is not based on the 8-flow directions

- Grid element
- Xcoord
- Ycoord
- Velresmax (fps or mps)
- Velxmax
- Velymax

*VELTIMEC.OUT*

This file lists the grid 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.

- Grid element
- Vmax in the channel element (fps or mps)
- Time of occurrence

*VELTIMEFP.OUT*

This file lists the first 100 floodplain elements: number, maximum floodplain 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 floodplain velocities.

- Grid element
- Vmax floodplain element (fps or mps)
- Time of occurrence

*VELTIMEST.OUT*

This file lists the street element number, maximum street 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 street velocities.

- Grid element
- Vmax street element (fps or mps)
- Time of occurrence

### *WSTIME.OUT*

If the 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).

- Grid element
- Flow area of the cross section (sqft or sqm)
- Top width of the cross section (ft or m)
- Wetted Perimeter of the cross section (ft or m)

### *XSEC.OUT*

This file is created by the channel sediment transport option (ISED = 1 in CONT. DAT and ISEDN = 1 for a channel segment in CHAN.DAT) for natural cross section geometry data. It contains the final cross section bed elevations after scour and deposition have been computed. The file looks the same as XSEC.DAT with updated elevation data.

# 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 file browser.

## 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

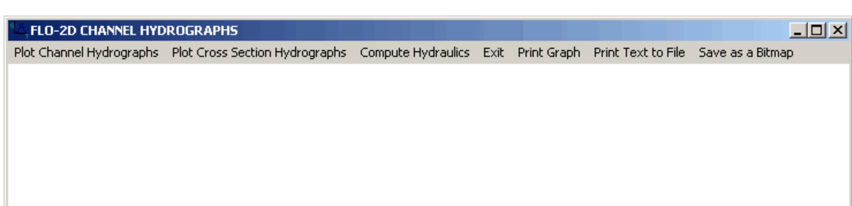
| | |
|---|---|
| 3 | Line 1: Number of gaging station with hydrograph data |
| Haynor | Line 2: Name of gaging station (10 letters) |
| 13160 251 | Line 3: Channel grid element and # of hydrograph pairs |
| 0.00 1287.85 | Line 4: Hydrograph pairs time(hours) discharge (cfs) |
| 1.00 1285.47 | Line 4: Hydrograph pairs time(hours) discharge (cfs) |

Notes:
Line 2 - 4: These lines are repeated for each gaging station.

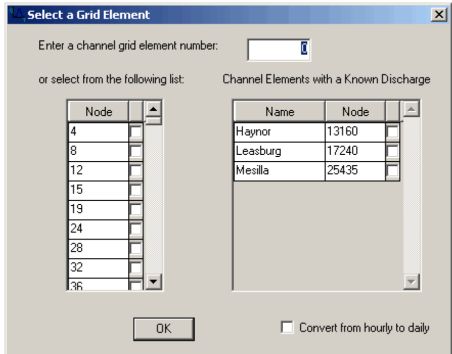
## HYDRO.DAT File Example

```
Haynor
13160   251
0.00    1287.85
1.00    1285.47
2.00    1295.01
3.05    1302.20
...
```

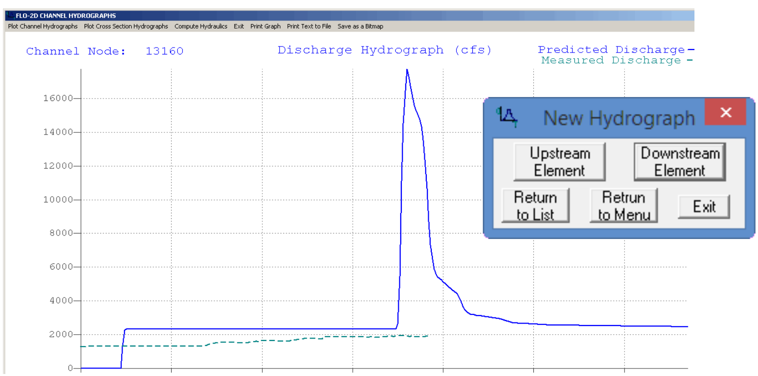
After opening HYDROG, click on either ‘Plot Channel Hydrographs’, ‘Plot Cross Section Hydrographs,’ or ‘Compute Hydraulics’ in the Main Menu shown below:



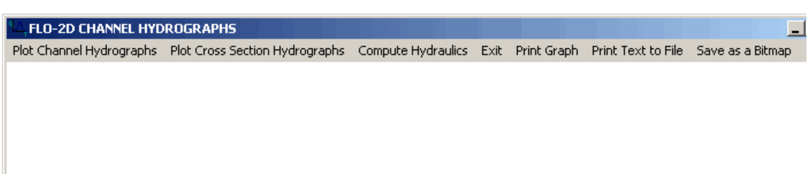
Click on 'Plot Channel Hydrographs' a dialog box appears to select either a channel segment or element:




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.



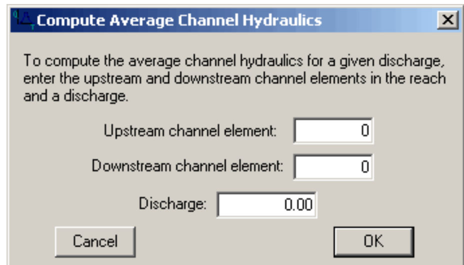

Similar hydrographs can plotted for floodplain cross section selected in the FPXSEC.DAT. If this file exists, then the cross section hydrograph for the selected cross section elements and flow direction will plotted using the second command on the Main Menu bar.



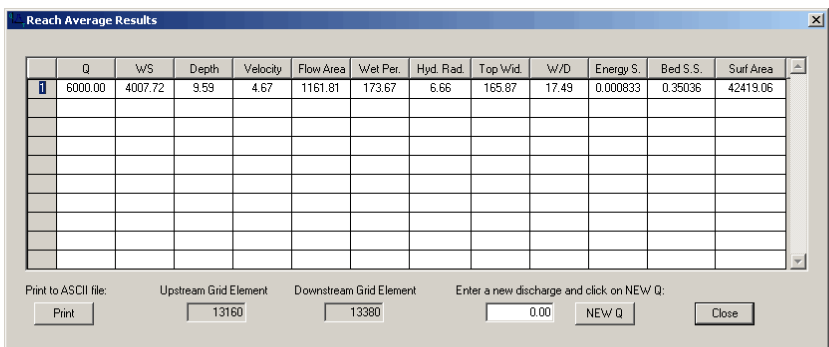

Hint:

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:




After entering 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:



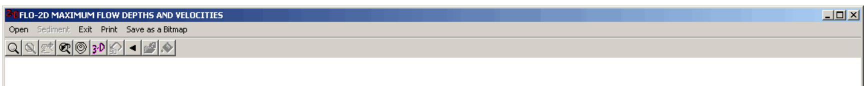

| | Q | WS | Depth | Velocity | Flow Area | Wet Per. | Hyd. Rad. | Top Wid. | W/D | Energy S. | Bed S.S. | Surf Area |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 6000.00 | 4007.72 | 9.59 | 4.67 | 1161.81 | 173.67 | 6.66 | 165.87 | 17.49 | 0.000833 | 0.35036 | 42419.06 |

This table displays the average discharge weighted hydraulic conditions for the given discharge between the two channel elements (inclusive). From this dialog box, select a new discharge and add to the table or print this table to an ASCII file (HYDR.OUT).

## 6.2 MAPPER PRO

MAPPER Pro 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 others.  MAPPER Pro 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.



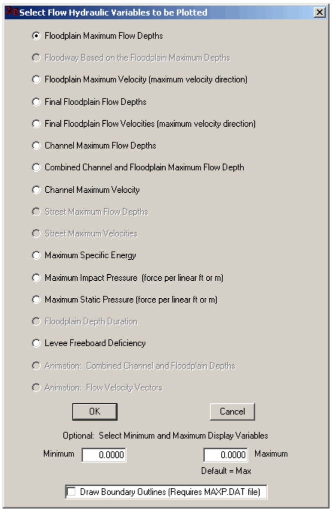

The MAPPER Pro manual describes the commands and tools and provides instruction.

## 6.3 MAXPLOT

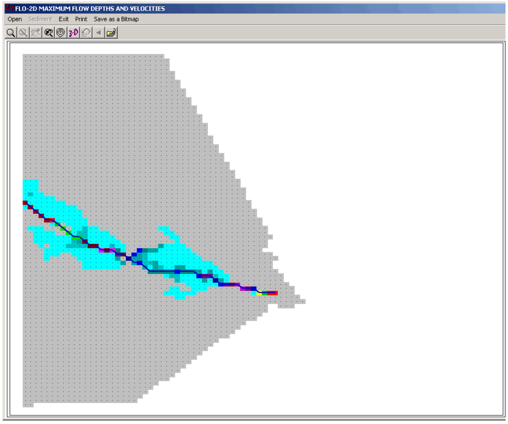
The MAXPLOT program is a basic graphical tool to display the grid element maximum depths and velocities.  MAXPLOT is a simple alternative to MAPPER Pro that quickly displays plots of the maximum floodplain and channel depths, maximum street velocity, final floodplain depths and others.  It is faster than MAPPER Pro but has less graphical resolution and fewer display options.  Use MAXPLOT for 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 clicking 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. 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 listing 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.

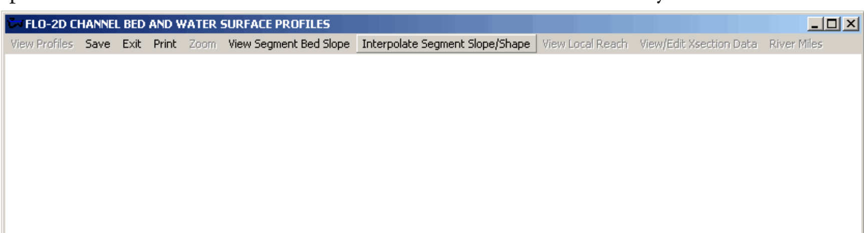
Line 2. Number of points in each boundary set.

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.

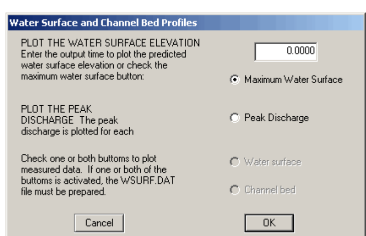
## 6.4 PROFILES

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

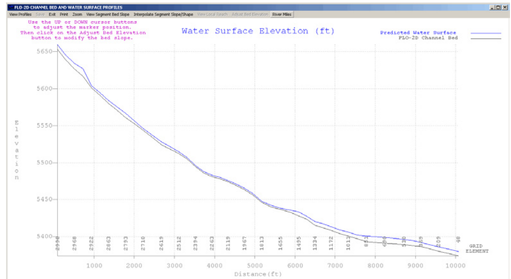



channel simulation first. The PROFILES program has zoom and print options to assist in reviewing the results.

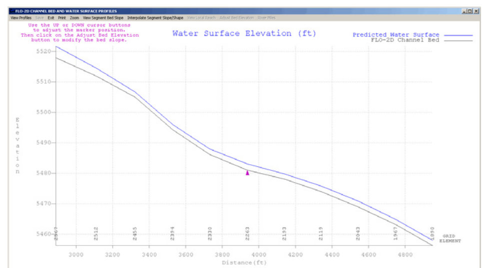
To view the predicted water surface profiles, click on 'View Profiles' in the Main Menu and a dialog box appears:




To view the predicted maximum water surface elevation profile, click on the radio button labeled 'Maximum Water Surface' and click 'OK'. 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. 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 and scour in a reach of the Middle Rio Grande in New Mexico.


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 the 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 Troubleshooting Guidelines

### Data Errors

Data input errors may result in the automatic termination of a simulation run along with a Fortran error message. The error message will report a 'Unit' number that is associated with the FLO-2D file that contains the error. The files are listed in Table 4.1, Table 7.1, and Table 7.2.

**Table 7.1 List of Misc Files and Unit Numbers**

| Unit No. | File Name | Unit No. | File Name |
|---|---|---|---|
| 47 | ARF.BAC | 398 | MANNINGS_N.BAC |
| 260 | BREACH.BAC | 2902 | MANNINGS_N_RES.BAC |
| 387 | BUILDING_COLLAPSE.BAC | 48 | MULT.BAC |
| 46 | CHAN.BAC | 51 | OUTFLOW.BAC |
| 40 | CONT.BAC | 42 | RAIN.BAC |
| 35 | EVAPOR.BAC | 1569 | SDCLOGGING.BAC |
| 41 | FPLAIN.BAC | 49 | SED.BAC |
| 121 | FPXSEC.BAC | 53 | STREET.BAC |
| 1610 | GUTTER.BAC | 1567 | SWMMFLO.BAC |
| 69 | HYSTRUC.BAC | 1564 | SWMMOUTF.BAC |
| 43 | INFIL.BAC | 29 | TOLER.BAC |
| 44 | INFLOW.BAC | 397 | TOPO.BAC |
| 58 | LEVEE.BAC | 2901 | TOPO_RES.BAC |

## 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 MANNINGS_M.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. MANNINGS_N.RGH = MANNINGS_N.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 output interval 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.

## 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 critical part of 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.

## 7.2 Trouble Shooting Technique

When undertaking a new FLO-2D flood simulation, start simple and progressively build in model component detail. After the required data files have been prepared, run a basic overland flood simulation. Review the results. If any issues arise consult the troubleshooting tips found in this chapter.

To debug the data files after a FLO-2D simulation, begin by reviewing the ERROR.CHK file. All the data errors recognized by the model are reported in this file. FLO-2D has an extensive data error and warning message system and the messages are reported in ERROR.CHK as data inconsistencies are encountered. One of the most common errors is missing data that will invoke an end-of-file error statement to the screen. 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 the component switch ICHANNEL = 1 in the CONT.DAT file, but the data file CHAN.DAT is not available.

One data error that is difficult to locate is the array allocation violation where the array index number becomes zero or larger than the assigned value. For example, there may be missing sediment concentrations in INFLOW.DAT for a mudflow simulation. This made a code error where a variable is not initialized to zero. When this type of error is encountered, the FLO-2D model is terminated with a FORTRAN 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 turn them back on one at time until the error occurs again. Reset simulation time to the model time just after the error occurred to reduce time to debug the model. If attempts to debug an error are ineffective, send a zipped copy of the data files to FLO-2D (contact@flo-2d.com) along with brief description of the problem.

The user can create a set of backup data files to debug the model. Set IBACKUP = 1 in the CONT.DAT file. These backup files replicate the 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, 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.

## 7.3 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 7.3. List of 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. |
| 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. |

## 7.4 Runtime Errors

The following guidelines are provided to assist the user in resolving 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. The problem grid element may also be listed in the TIME.OUT file with the highest number of timestep decrements 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 for a natural channel (not a concrete rectangular or trapezoidal channel geometry) 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. Use PROFILES to re-interpolate between surveyed cross sections.
- Review and adjust the bed slope with the PROFILES program. Adverse bed slopes are OK but adverse spikes and dips are not.
- Select a longer channel length within the channel grid element.

## 7.5 Debugging Errors

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 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 or in ERROR.CHK:

- 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. Check the contiguous grid elements to the problem element in the 8 directions as the problem may be with the neighbor element.

*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 the model may still have relatively good volume conservation.

*Supercritical flow*

*S*upercritical 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 a reasonable lower limit of 0.2 is reached.
- 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 or the next one upstream.

*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 and check to see that it 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 TOPO.DAT or in the GDS.
- If the depressed element is a gravel pit or some other feature, increase the n-value to decrease the velocity (vertical overfall 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 in reaches that are expanding and contracting causing rapidly variable storage. During high flow in a large river, the variation in discharge associated with stage change on the order of ~0.2 ft can be 1,000 cfs or more. Review the numerical surging troubleshooting. If the channel surging is severe, the 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 TOPO.DAT file. This type of error generally occurs when the user edits the TOPO.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. Multiple left bank elements can be assigned to the same right bank on a river bend. If a channel extends through a right bank element, the model will generate 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 bank alignment, 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.

*Program stops; identifying one or more grid elements with too little floodplain surface.*

The model will generate a message 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 errors are reported in this file, delete CADPTS.DAT and FPLAIN.DAT and run the model again. The FLOPRO.EXE will rewrite this file.

## 7.6 Debug Output Tables

The DEBUG.OUT file is created when the user runs the model in Debug model via the QGIS Plugin. The error codes in Tables 7.4, 7.5, and 7.6 are the codes used in the Debug system. They help identify data errors and data conflicts. These files are generated as part of the preliminary data checks. These error checks do not include any simulation results. Table 7.5 and 7.6 offer basic corrective actions for the errors.

**Table 7.4. ERROR CODE CATEGORIES**

| Error Code | Error Category |
|---|---|
| 100 | Switches, Control Variables, Version |
| 200 | Boundary, Coordinate, Floodplain, Elevation |
| 300 | Stability Criteria |
| 400 | TOL |
| 500 | Roughness |
| 600 | Rainfall |
| 700 | Infiltration |

| Table 7.4. ERROR CODE CATEGORIES | |
|---|---|
| 800 | Inflow, Outflow |
| 1000 | Channel |
| 2000 | Hydraulic Structures |
| 3000 | Streets, ARF/WRF |
| 4000 | Storm Drain |
| 5000 | Cross Sections |
| 6000 | Sediment, Mud |
| 7000 | Levees |
| 8000 | Multiple Channels |

| Table 7.5. BASIC ERROR CODES | | |
|---|---|---|
| 100 | Versions of the FLO-2D Pro and Storm Drain are Different. Please Check FLO-2D Build and Update Vc2005-Con.Dll | |
| 100 | Variable ICHNSEG must be Greater Than 0 | |
| 100 | Floodway Switch = 1,Set Encroach in CONT.DAT | |
| 100 | Set NOPRTC to Only 0, 1, or 2 in CONT.DAT | |
| 100 | For Graphical Display (Lgplot=2),Graptim must be Greater Than 0 | |
| 100 | Variable Xconc Exceeds 1 | |
| 100 | Variable Xarf is Less Than 0 or Greater Than 1 | |
| 100 | Variable Froudl Greater Than 9 | |
| 100 | Variable Noprtfp is a Switch,Use Only 0,1,2 or 3 | |
| 100 | Mudflow (Mud=1) and Conventional Sediment Transport (Ised=1) Cannot Be Modeled in the Same Simulation. Review CONT.DAT File | |
| 100 | Grid Element 1 Has No Neighbor Grid Elements,Check the CADPTS.DAT File | |
| 100 | If Displaying the Flood Graphics - Lgplot = 2 in CONT.DAT - Then Ideplt must be Greater Than Zero in INFLOW.DAT | |
| 100 | If Only Writing Text Output to Screen - No Flood Graphics Lgplot = 0 in CONT.DAT - Set Ideplt = 0 in INFLOW.DAT | |
| 100 | Ideplt (INFLOW.DAT) must be an Inflow Node and the CONT.DAT Variable Lgplot must be Set to 1 | |
| 100 | Total Simulation Time of the Model Exceeds the Hydrograph Duration | |
| 100 | If Ideplt is Listed As Inflow Node in the INFLOW.DAT File,Then Lgplot must be 0 or 1 | |

| | |
|---|---|
| | Review engine file dates and flopro.exe and vc2005con.dll. Make sure the file dates correspond to builds that are the same. This may require Technical Support. |
| | I can't find this variable. |
| | To run a floodway simulation, set Floodway Switch = 1 and set the Encroach variable in CONT.DAT. |
| | NOPRTC is a switch. The positions are 0, 1 or 2. |
| | The variable Graphtim is missing in CONT.DAT. |
| | The sediment concentration cannot be greater than 1. |
| | The Xarf variable must be a value between 0 and 1. |
| | The Froudl variable should not be greater than 1. |
| | NOPRTFP is a switch. The positions are 0, 1 or 2. |
| | Set either MUD or ISED to 0. |
| | If grid element number 1 does not have a neighbor, it is dangling or the coordinates are wrong in TOPO.DAT. Check the location of the cell. Correct it by realigning the grid to the computational domain. |
| | Set ideplt to an inflow grid element number in inflow.dat. |
| | For text mode, set lgplot = 0 and ideplt = 0. |
| | Make sure Ideplt is a grid elment listed in inflow.dat. |
| | If the hydrograph ends before the simulation, make sure it is set to zero or the last discharge in the hydrograph will continue as steady flow. |
| | Turn on the Lgplot and Graphtim to use Display Mode. |

| Table 7.5. BASIC ERROR CODES | | |
|---|---|---|
| 200 | Grid Element Coordinates Exceed 1000000000. Reduce the Coordinate Values Before Proceeding | |
| 200 | Hydraulic Structure Channel Inflow must be a Channel Element | |
| 200 | Time-Stage Elements Have a Stage Assigned that Was Less Than the Floodplain or Channel Bed Elevation. Stage Was Reset to the Bed Elevation | |
| 200 | If Ideplt is 0 in INFLOW.DAT and Irain is 0 in CONT.DAT,There is No Inflow to Be Plotted. | |
| 300 | A Channel/Street Courant Number is Required in TOLER.DAT | |
| 300 | If Istrflo in STREET.DAT is Set to 1,Then at Least One Inflow Node Must Have a Street in It | |
| 400 | Variable Tol Has an Inappropriate Value | |
| 400 | Please Review If Tol = 0.05 Ft or 0.015 M With the Rainfall Abstraction | |
| 500 | MANNINGS_N.DAT File Has a Mismatched Grid Element Number...Check the End of this File | |
| 500 | MANNINGS_N.DAT Files Does Not Exist. Create the File Before Proceeding | |
| 500 | The Spatially Variable Shallown Value is Outside the Range 0.010 to 0.99 | |
| 500 | N-Value is Less Than 0 or Greater Than 1 | |
| 600 | Line 2 in RAIN.DAT File Has to Be Reviewed For Spatially Variable Real Rainfall Adjustments (Irainarf=1) With Rainarf Values | |
| 600 | Rtt must be Greater Than 0 | |
| 600 | First Pair of the Rainfall Distribution Should Be 0. 0. | |
| 600 | Date and Time in Raincell.Dat Must Have this Format: 06-15-2003 14:00:00 | |
| 700 | Variable Infmethod Line 1 in the INFIL.DAT is Either Missing or Not Correctly Assigned | |
| 700 | To Use the Scs Curve Number Method For Infiltration You Must Have Rainfall,Irain = 1 in CONT.DAT and RAIN.DAT File | |

| | |
|---|---|
| | Check the coordinates in topo.dat. |
| | Reposition the structure node onto a left bank node. |
| | Check the invert elevation of the structure, the grid element elevation or the head reference elevation. |
| | Either Set Lgplot = 0,Assign Ideplt an Inflow Hydrograph in INFLOW.DAT,Or Set Irain =1 in CONT.DAT and Assign the RAIN.DAT File |
| | Set the correct Courant number. |
| | Check the STREET.DAT file. |
| | Check the TOL value.  It must be in a corect rangew. |
| | Check the TOL variable and the Initial Abtraction variable.  The initial abstraction may be too high.  See INFIL.DAT. |
| | The MANNINGS_N.DAT file might not be complete. |
| | Export MANINGS_N.DAT again. |
| | Check the SPATIALSHALLOWN.DAT file. |
| | Check the CONT.DAT file. |
| | Spatially variable data is missing.  Check RAIN.DAT. |
| | Check RAIN.DAT. |
| | Correct the first data pair of the rainfall distribution curve.  Set the first data pair to 0.0 0.0. |
| | Check RAINCELL.DAT. |
| | Check INFIL.DAT. |
| | Check RAIN.DAT. |

| TABLE 7.5. BASIC ERROR CODES | | |
|---|---|---|
| 700 | Variable Poros is Greater Than 1 | |
| 700 | Variable Sati or Satf is Greater Than 1 | |
| 700 | Variable Rtimpf Exceeds 1.0. Do Not Enter As a Percent Use a Fraction | |
| 700 | Abstraction Exceeds the Total Rainfall (Impossible) For at Least One Grid Element and May Result in Volume Conservation Error | |
| 700 | Initial Abstraction > Tol (Depression Storage). Consider (Not Required) Lowering the Tol Value or Adjusting the Ia Value | |
| 800 | There are Two Inflow Conditions Imposed at the Same Cell | |
| 800 | This Grid Cell Has an Inflow and a Full ARF | |
| 800 | This Grid Cell Has an Inflow and a Partial ARF | |
| 800 | The Following Cell Has an Inflow and a Hs | |
| 800 | The Following Cell Has an Inflow Fp on a Channel Left Bank Element | |
| 800 | The Following Cell Has an Inflow Fp on a Channel Right Bank Element | |
| 800 | There are an Inflow Conditions Imposed on a Levee Element | |
| 800 | This Grid Cell Has an Inflow on a Multiple Ch Element | |
| 800 | This Grid Cell Has an Inflow on a Multiple Ch Element | |
| 800 | There are Two Inflow Conditions Imposed at the Same Cell | |
| 800 | The Following Cell Has an Inflow Ch on a Channel Right Bank Element | |
| 800 | There are an Inflow Conditions Imposed on a Levee Element | |
| 800 | There are Two Outflow Conditions Imposed at the Same Cell | |
| 800 | The Following Cell Has a Channel Outflow on a Channel Rigth Bank Element | |
| 800 | There are an Outflow Conditions Imposed on a Levee Element | |
| 800 | There are Two Outflow Conditions Imposed at the Same Cell | |
| 800 | The Following Cell Has an Outflow (Fp) on a Channel Left Bank or Rigth Bank Element: | |

| | |
|---|---|
| | Check INFIL.DAT. |
| | Check INFIL.DAT. |
| | Check INFIL.DAT. |
| | Check spatial abstraction variable in INFIL.DAT. |
| | The TOL variable and IA variable can be summed to account for the initial abstraction. |
| | A cell is listed twice in INFLOW.DAT. Check the file and remove one of the hydrgraphs. |
| | Reposition the inflow node. |
| | Consider repositioning the inflow node. |
| | Repostion the inflow node or the hydraulic structure inlet node. |
| | Consider changing the inflow to channel inflow. |
| | Consider moving the inflow node to the left bank and changing it to a channel node. |
| | Check the levee Inflow condition. Make sure the inflow is on the correct side of the levee and make sure the cell elevtion is set correctly. |
| | Reposition the inflow node. |
| | Reposition the inflow node. |
| | A cell is listed twice in INFLOW.DAT. Check the file and remove one of the hydrgraphs. |
| | Move the inflow node to the left bank. |
| | Check the levee Inflow condition. Make sure the inflow is on the correct side of the levee and make sure the cell elevtion is set correctly. |
| | Remove the extra line in OUTFLOW.DAT. |
| | Move the outflow node left bank. |
| | Make sure the outflow node is on the correct side of the levee. |
| | Move the outflow node left bank. |
| | It's OK for n FP outflow node to be on a left bank but not a right bank. |

| Table 7.5. BASIC ERROR CODES | | |
|---|---|---|
| 800 | There are an Outflow Conditions Imposed on a Levee Element | |
| 800 | There are Two Stage Time Relationships Imposed at the Same Cell | |
| 800 | The Following Cell Has Stage Time Relationship on a Channel Right Bank Element: | |
| 800 | There are a Stage Time Outflow Condition Imposed on a Levee Element | |
| 800 | There are a Stage Time Relationship Imposed on an Outflow Cell | |
| 800 | There are a Floodplain Outflow and a Stage Time Relationship at the Same Cell | |
| 800 | There are Two Outflow Conditions Imposed at the Same Cell | |
| 800 | This Grid Cell Has an Outflow and a Full ARF | |
| 800 | This Grid Cell Has an Outflow and a Partial ARF | |
| 800 | The Following Cell Has an Outflow and a WRF: | |
| 800 | This Grid Cell Has a Stage Time Relationship and a Full ARF | |
| 800 | This Grid Cell Has a Stage Time Relationship and a Partial ARF | |
| 800 | The Following Cell Has an Outflow and a WRF: | |
| 800 | This Grid Cell Has an Outflow and a Full ARF | |
| 800 | This Grid Cell Has an Outflow and a Partial ARF | |
| 800 | The Following Cell Has an Outflow and a WRF: | |
| 800 | An Inflow Hydrograph Has Been Assigned to a Channel Element (C-Line in INFLOW.DAT) and There is No Channel Component (Ichannel = 0 in CONT.DAT) | |
| 800 | First Pair of the Floodplain Hydrograph Should Be 0. 0. to Interpolate the First Timestep | |
| 800 | No Inflow Discharge Specified For the Inflow Element | |
| 800 | INFLOW.DAT Variable Ideplt must be an Inflow Node and an Inflow Node - Khin - Variable in INFLOW.DAT must be Specified, CONT.DAT Variable Inplot must be Set to 1 | |

| | |
|---|---|
| | Make sure the outflow node is on the correct side of the levee. |
| | Remove one of the duplicate stage time conditions from OUTFLOW.DAT. |
| | Remove the ouflow from the right bank. |
| | Make sure the outflow node is on the correct side of the levee. |
| | |
| | |
| | Delete one of the outflow nodes in OUTFLOW.DAT. |
| | Delete the outflow node or the ARF. |
| | Delete the ARF. |
| | Delete the WRF. |
| | Delete the outflow node or the ARF. |
| | Delete the ARF. |
| | Delete the WRF. |
| | Delete the outflow node or the ARF. |
| | Delete the ARF. |
| | Delete the WRF. |
| | Turn the channel switch on or reset the inflow node to floodplain. |
| | Set the first data pair to 0.0 0.0 in the INFLOW.DAT. |
| | Check INFLOW.DAT. |
| | To run in display mode, set the graphics mode in CONT.DAT and the plotting hydrograph in INFLOW.DAT. |

| TABLE 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 1000 | Inflow Fp on a Ch Interior Element | |
| 1000 | Inflow Ch on a Ch Interior Element | |
| 1000 | Outflow Ch on a Ch Interior Element | |
| 1000 | Outflow Fp on a Ch Interior Element | |
| 1000 | Stage Time Relationship on a Ch Interior Element | |
| 1000 | Full ARF on a Ch Interior Element | |
| 1000 | Partial ARF on a Ch Interior Element | |
| 1000 | WRF on a Ch Interior Element | |
| 1000 | Hs inlet on a Ch Interior Element | |
| 1000 | Hs outlet on a Ch Interior Element | |
| 1000 | Levee on a Ch Interior Element | |
| 1000 | Multiple Channel on a Channel Interior Element | |
| 1000 | Channel Width is Greater Than the Element Width. Channel Left and Right Bank Elements Should Be Separated | |
| 1000 | Channel Grid Element Will Require Separate Left and Right Bank Elements | |
| 1000 | Channel Extension Exceeds the Grid System Boundary | |
| 1000 | Channel Element Extends Into Interior of the Channel Element Instead Extend the Channel Into Another Bank Element | |
| 1000 | Channel Element is Repeated in the CHAN.DAT File. Each Channel Element Should Only Be Listed Once | |
| 1000 | Channel Right Bank Elements Need Some Adjustment Due to the Channel Width. Set Right Bank Either Closer or Farther Away from the Left Bank Element | |
| 1000 | Remaining Floodplain Surface Area on the Channel Bank Elements Needs to Be Larger For Left Bank Element | |
| 1000 | Data Error...Check the Channel Elements in the CHAN.DAT Files | |
| 1000 | Channel Extension For Grid Element Extends Into Another Channel Element | |

| | |
|---|---|
| | Move inflow node or realign channel. |
| | Move inflow node or realign channel. |
| | Move outflow node or realign channel. |
| | Move outflow node or realign channel. |
| | Move outflow node or realign channel. |
| | Delete ARF or realign channel. |
| | Delete ARF or realign channel. |
| | Delete WRF or realign channel. |
| | Move hydraulic structure or realign channel. |
| | Move hydraulic structure of realing channel. |
| | Realign levee or realign channel. |
| | Realign multiple channel. See reference manual. |
| | Realign right bank. Extend right bank way from left bank. |
| | Realign right bank. |
| | Realign right bank. |
| | Realign right bank. |
| | Eliminate one of the repeated channel elements. Tributary and Split flows should connect along adjacent banks. |
| | Realign right bank. |
| | Extend right bank away from left bank. |
| | Review CHAN.DAT. Load project in PROFILES.EXE to troubleshoot. |
| | Ralign right bank. |

| Table 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 1000 | Channel Confluence Element Does Not Have Enough Connections, or a Channel Segment is Beginning or Ending at a Main Channel Confluence Element | |
| 1000 | Channel Extends Past the Levee System, Please Review the CHANNEL. CHK File and Make the Necessary Corrections | |
| 1000 | Inflow Channel Element is not a Channel Element in CHAN.DAT | |
| 1000 | Channel Outflow Node Must Have a Lower Bed Elevation Than the Contiguous Upstream Channel Element to Compute a Normal Depth Outflow Condition | |
| 1000 | Channel Outflow Variable - Kout - in the OUTFLOW.DAT File must be a Channel Element in the CHAN.DAT File | |
| 2000 | This Grid Cell Has a Hs Inlet and a Full ARF | |
| 2000 | This Grid Cell Has a Hs Outlet and a Full ARF | |
| 2000 | This Grid Cell Has a Hs Inlet and a Partial ARF | |
| 2000 | This Grid Cell Has a Hs Outlet and a Partial ARF | |
| 2000 | This Grid Cell Has a Hs on a Channel Rb Element | |
| 2000 | Inlet on a Full ARF Element | |
| 2000 | Hydraulic Structure Has an Adverse Bed Slope. Outlet Invert is Higher Than the Inlet Invert. Please Check to Ensure this is Correct | |
| 2000 | Hydraulic Structure Has a Reference Elevation that is Lower Than the Inlet Node Bed Elevation | |
| 2000 | Hydraulic Structure Has an Inflow or Outflow Element that is Not a Channel | |
| 2000 | Hydraulic Structure Has a Name Length Longer Than 30 Characters. | |
| 2000 | A Hydraulic Structure Has Been Assigned to a Channel Element. Channel is turned off. | |
| 2000 | Hydraulic Structure Rating Curve, Rating Table, Or Generalized Culvert Switch (Icurvtable) Does Not Match the Assigned Data | |

| | |
|---|---|
| | Review confluence elements. The tributary or split channel may not be close enough to the main channel banks. |
| | Realign the channel or the levee. |
| | Move inflow node to a left bank or reset the node to floodplain or turn the channel switch on. |
| | Review the channel invert elevation and make the necessary correction so that theoutflow node can calulate normal depth. The outflow invert elevation must be lower than that of the upstream node. |
| | Move the outflow node to a left bank, reset the node to floodplain or turn the channel switch on. |
| | Move the hydraulic structure node. |
| | Move the hydraulic structure node. |
| | Move the hydraulic structure node or reset the ARF to zero. |
| | Move the hydraulic structure node or reset the ARF to zero. |
| | Move the hydraulic structure to the left bank or change it to a floodplain structure. |
| | Move Inlet |
| | Review inver elevations. Apply elevation corrections if necessary. Validate structure direction. |
| | Correct invert elevation or correct head reference elevation or set head reference elevation to zero. |
| | Move inlet node to the channel bank or change it to a floodplain structure. |
| | Shorten the Name to Less Than 30 Characters |
| | (Ifporchan > 0 line S in HYSTRUC.DAT) and there is no channel component (Ichannel = 0 in CONT.DAT). Turn on channel switch. |
| | Review HYSTRUC.DAT and set the switch to the correct position to match the assigned data. |

| Table 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 2000 | Hydraulic Structure must have a Culvert Area Coefficient and Exponent For Routing in a Long Culvert. | |
| 2000 | Make Sure that the "Atable" Variable on Line 4 of the HYSTRUC.DAT File is Included | |
| 2000 | First Data Pair of a Hydraulic Structure Rating Table Should Be 0. 0. to Interpolate the Next Data Pair | |
| 2000 | Hydraulic Structure Rating Curve Stage Must Increase With Increasing Discharge | |
| 2000 | Rate of Change in the Following Hydraulic Structure Rating Tables May Be Unreasonable - Rate of Change = 10 Times Previous Stage Rate of Change | |
| 2000 | If the Generalized Culvert Equations are Being Used. The Inoutcont Tailwater Control is Not Necessary. Set Inoutcont = 0 | |
| 2000 | Culvert Length Must Assign in the S-Line of the HYSTRUC.DAT If the Generalized Culvert Equations are Being Used | |
| 2000 | Hydraulic Structure Inflow Node is Repeated More Than Once | |
| 2000 | Hydraulic Structure Outflow Node is Repeated More Than Once Without Assigning a D-Line Conveyance Capacity Limitation. | |
| 2000 | Hydraulic Structure Has a Reference Elevation that is Lower Than the Inflow Node Bed Elevation | |
| 2000 | Hydraulic Structure Channel Outflow must be a Channel Element | |
| 2000 | Hydraulic Structure Has a Reference Elevation that is Lower Than the Inflow Node Bed Elevation | |
| 2000 | Hydraulic Structure Channel Inflow Element must be a Channel Element | |
| 2000 | Hydraulic Structure Inflow Element Cannot Be a Grid System Outflow Element | |

| | |
|---|---|
| | The clength and cdiameter was assigned, assign the culvert area coefficient and exponent so FLO-2D can simulate the culvert volume and travel time. |
| | This table is required if clength and cdiameter are used in a Rating Table structure. |
| | Reset first row of table data to 0.00 0.00. |
| | The rating curve data has an error. Check the data so the discharge increases with increasing stage. |
| | Check the rating table. It may require more data pairs or it may be incorrect. |
| | Set inoutcont to 0. |
| | Assign culvert length and depth in the S line. |
| | Review HYSTRUC.DAT. Make sure each inflow node is only listed once. If two nodes are near each other, separate them by a grid elment. |
| | Review HYSTRUC.DAT. Make sure each outflow node is only listed once. If two nodes are near each other, separate them by a grid elment. |
| | Correct invert elevation or correct head reference elevation or set head reference elevation to zero. |
| | Check the position of the outlet element or make sure the channel switch is on in CONT.DAT. |
| | Correct invert elevation or correct head reference elevation or set head reference elevation to zero. |
| | Check the position of the outlet element or make sure the channel switch is on in CONT.DAT. |
| | Correct invert elevation or correct head reference elevation or set head reference elevation to zero. |

| Table 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 2000 | Hydraulic Structure Outflow Element Cannot Be a Grid System Outflow Element | |
| 3000 | The Following Cell Has a Full ARF on a Channel Left or Right Bank Element | |
| 3000 | The Following Cell Has a Partial ARF on a Channel Left or Right Bank Element | |
| 3000 | Street on an Outfall Element | |
| 3000 | Full ARF on a 1D Street | |
| 3000 | Partial ARF on a 1D Street | |
| 3000 | Hs Inlet on a 1D Street | |
| 3000 | Hs Outlet on a 1D Street | |
| 3000 | Multiple Channel on a 1D Street | |
| 3000 | Gutter on a 1D Street | |
| 3000 | Variable Strman is Less Than 0 or Greater Than 1 | |
| 3000 | Variable Istrflo is a Switch, Use Only 0 or 1 | |
| 3000 | Variable Depx must be Greater Than 0 | |
| 3000 | Variable Widst must be Greater Than 0 | |
| 3000 | Variable Igridn must be Greater Than 0 | |
| 3000 | Grid Elements are Defined More Than Once (Street.Dat) For a Street Intersection Within a Grid Element | |
| 3000 | Street Elements (Street.Dat) are Missing Line "W" in the Street.Dat File | |
| 3000 | Variable Istdir must be Greater Than 0 and Less Than or Equal to 8 | |
| 3000 | Variable Widr must be Greater Than 0 | |
| 3000 | Grid Element ARF Values Were Adjusted | |

| |
|---|
| Move the outlet element to a node that is adjacent to the outflow node. |
| Realign the channel or eliminate the ARF. |
| Delete the ARF. |
| I don't know how to fix this. |
| Realign street or delete ARF. |
| Delete ARF. |
| Move hydraulic structure or realign street. |
| Move hydraulic structure or realign street. |
| Reposition multiple channel nodes or realign street. |
| Delete gutter or delete street. |
| Assign street Manning's N correctly. |
| Apply variable correctly. |
| Assing street depth. |
| Assign street width. |
| Assing correct Manning's n value. |
| Delete one of the misassigned street elements. |
| W lines are necessary to define the street direction in the cell. Assign them as shown in Lesson 11. |
| Add correct street direction. |
| Correct street width. |
| See ARF.DAT for automatic correction list. ARFs were reasigned 1.0 to Eliminate the Potential For Instability Related to Small Surface Area. These are Reported to the ARF_Adjustment.Chk File |

| Table 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 3000 | Impervious Area Represented By the Rtimp Percentage is Less Than the ARF Value For at Least One Grid Element | |
| 3000 | A Channel Element Has One or More Street Segments. Remove the Street Segments from this Element | |
| 4000 | Inlet on a Full ARF Element | |
| 4000 | Inlet on a Partial ARF Element | |
| 4000 | Outfall on a Full ARF Element | |
| 4000 | Outfall on a Partial ARF Element | |
| 4000 | Outfall on a Levee Element | |
| 4000 | Inlet on a Levee Element | |
| 4000 | Duplicate Inlet on SWMMFLO.DAT | |
| 4000 | Inlet on an Outfall | |
| 4000 | Outfall on an Outfall | |
| 4000 | Channel Rb on a Inlet Element | |
| 4000 | Channel Rb on an Outfall Element | |
| 4000 | Multiple Channel on a Inlet Element | |
| 4000 | Multiple Channel on an Outfall Element | |
| 4000 | There is a Levee and a Storm Drain Inlet Assigned to Grid Cell | |
| 4000 | There is a Storm Drain Inlet Assigned to Completely Blocked Grid Cell | |
| 4000 | There is a Storm Drain Outfall Assigned to Completely Blocked Grid Cell | |
| 4000 | There is a Hydraulic Structure and a Storm Drain Inlet Assigned to Grid Cell | |
| 4000 | Storm Drain Inlet Has Invert Elevation Errors. Please Check Invert Elevation and Rim Elevation For Node | |
| 4000 | Curb Opening Height must be Greater Than Zero. Please Revise SWMMF-LO.DAT File | |

| | |
|---|---|
| | Impervious area should represent the building blockage and any other potential impervous area. It should be at least the same as the ARF value. |
| | Realign the street or channel. Review aerial images to assign channel or street alignment. |
| | Move Inlet. |
| | Move Inlet. |
| | Move Outfall or delete ARF. |
| | Move Oufall or delete ARF. |
| | Review outfall position. Make sure it is on the correct side of the levee. Review elevation. |
| | Make sure the inlet is on the correct side of the levee. Check the elevation of the cell so that it matches he rim elevaiton of the inlet or the invert elevation of the type 4. |
| | Delete the repeated inlet. |
| | Reposition the inlet or the outfall. |
| | Repostion one of the outfall nodes. |
| | Move the inlet to the left bank. |
| | Move the outfall to the left bank. |
| | Reposition the inlet or the multiple channel. |
| | Reposition the outfall or the multiple channel. |
| | Make sure the inlet is on the correct side of the levee. Check the elevation of the cell so that it matches he rim elevaiton of the inlet or the invert elevation of the type 4. |
| | Move the inlet or delete the ARF. |
| | Move the outfall or delete the ARF. |
| | Repostion the hydraulic strucure or the inlet. |
| | Do you mean Max Depth? |
| | Review SWMMFLOW.DAT. |

| TABLE 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 4000 | Length must be Greater Than Zero | |
| 4000 | Height must be Greater Than Zero | |
| 4000 | Typical Weir Drain Coefficient: Range 2.8 to 3.2 | |
| 4000 | Width or Height must be Greater Than Zero | |
| 4000 | Typical Weir Drain Coefficient: 2.3 | |
| 4000 | Perimeter must be Greater Than Zero | |
| 4000 | Area must be Greater Than Zero | |
| 4000 | Surcharge Depth must be Greater Than Zero | |
| 4000 | There is a Conflict Between Inlets in the SWMMFLO.DAT File and Sub-catchments in the SWMM.INP,Features in Both Lists Need to Be in the Same Order | |
| 4000 | Inlets in the SWMMFLO.DAT File must be Identical to the Listed Inlets Junction Table of SWMM.INP File | |
| 4000 | Multiple Inlets Assigned to One Grid Cell | |
| 4000 | There is a Type 4 Inlet (Review SWMMFLO.DAT File) that is Missing the Rating Table in the SWMMFLORT.DAT File | |
| 4000 | There is an Inflow Node and a Storm Drain Inlet Assigned to Grid Cell | |
| 4000 | There is an Inflow Node and a Storm Drain Outfall Assigned to Grid Cell | |
| 4000 | There is an Outflow Node and a Storm Drain Inlet Assigned to Grid Cell | |
| 4000 | There is an Outflow Node and a Storm Drain Outfall Assigned to Grid Cell | |
| 4000 | Storm Drain Outfall Nodes are in Channel Interior Elements, Re-Assign to the Channel Elements in CHAN.DAT | |
| 5000 | Cross Section Element Can Only Be Assigned Once in the FPXSEC.DAT File. | |
| 6000 | Variable Xconc Should Not Be Assigned If Mudflow With a Sediment Concentration is Assigned to the Inflow Hydrograph | |
| 6000 | No Sediment Data in the SED.DAT File | |

| |
|---|
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Review SWMMFLOW.DAT. |
| Check the order of the inlets and the subcatchments. |
| Check the order of the inlets in SWMMFLOW.DAT and SWMM.INP. |
| Reposition the inlet or delete it if it is a repeated line. |
| Add the table to SWMMFLOWRT.DAT. |
| Reposition the inflow node or the inlet. |
| Reposition the inflow node or the outfall. |
| Respostion the inlet. |
| Reposition the outfall or delete the outlet. |
| Reposition the nodes to the left bank or reassign then grid element in SWMMFLO.DAT. |
| Remove repeated grid elements in FPXSEC.DAT. If the Cross Section Includes the Channel Use Only the Left Bank Channel Element in CHAN.DAT |
| Do not assign Xconc in CONT.DAT. |
| Check the SED.DAT file. |

| Table 7.6. ADVANCED ERROR CODES | | |
|---|---|---|
| 6000 | Error in Line 1 (M-Line) of the SED.DAT File | |
| 6000 | Dry Weight of Sediment is Zero in the SED.DAT File and Thus the Porosity is Also Zero | |
| 6000 | Sediment Size Exceeds the Recommended Value For the Application of the Yang Equation | |
| 6000 | Error in Line 2 (S-Line) of the SED.DAT File | |
| 6000 | Error in Z-Line of the SED.DAT File | |
| 6000 | Error in P-Line of the SED.DAT File | |
| 6000 | Error in D-Line of the SED.DAT File | |
| 6000 | Scourdep Variable in SED.DAT Line E Should Be Positive (>0.) | |
| 6000 | Error in E-Line of the SED.DAT File | |
| 6000 | Error in R-Line of the SED.DAT File | |
| 6000 | Error in S-Line of the SED.DAT File | |
| 6000 | Error in N-Line of the SED.DAT File | |
| 6000 | Isedn variable is incorrect. | |
| 7000 | There are a Levee Element on a Complete Blocked Element | |
| 7000 | There are a Levee Element on a Partial Blocked Element | |
| 7000 | There are a Levee Element With a WRF | |
| 7000 | This Grid Cell Has a Hs Inlet on a Levee Element | |
| 7000 | This Grid Cell Has a Hs Outlet on a Levee Element | |
| 7000 | This Grid Cell Has Two Levees | |
| 8000 | This Grid Cell Has an Inflow on a Multiple Ch Element | |
| 8000 | This Grid Cell Has an Inflow on a Multiple Ch Element | |

| |
|---|
| Check the SED.DAT file for missing or incorrect mudflow data. |
| Set the Dry Weight variable in SED.DAT. |
| Check the sediment size fractions in SED.DAT. |
| Check the sediment transport data in SED.DAT. |
| Check the sediment transport equation, bed thickness or volumetric concentration. |
| Check the sediment diameter and percentage. |
| Check the debris basin volume and the debris grid element number. |
| Check the scour depth. |
| Check the scour depth. |
| Check the grid element numbers or position in the rigid bed cells. |
| Check the sediment supply coefficient and exponent. |
| Check the size distribution for sediment supply. |
| Isedn Variable Must Equal One of the Sediment Size Fraction Groups in SED.DAT that is Associated With a Sediment Transport Equation. Do Not Assign Isedn to a Sediment Transport Equation Number |
| Consider repositioning or deleting the levee. |
| Make sure the levee is on the correct side of the cell. |
| Make sure the levee and WRF relationship is correct. |
| Make sure the hydralic structure is on the correct side of the levee. Review the grid element elevation so that the water can get to and from the structure inlet and outlet nodes. |
| Make sure the hydralic structure is on the correct side of the levee. Review the grid element elevation so that the water can get to and from the structure inlet and outlet nodes. |
| Delete the repeaded levee. |
| Move the inflow node. |
| Move the inflow node. |

**Table 7.6. ADVANCED ERROR CODES**

| | | |
|---|---|---|
| 8000 | This Grid Cell Has an Inflow on a Multiple Ch Element | |
| 8000 | This Grid Cell Has a Full/Partial ARF or WRF on a Multiple Ch Element | |
| 8000 | This Grid Cell Has a Full/Partial ARF or WRF on a Multiple Ch Element | |
| 8000 | This Grid Cell Has a Full/Partial ARF or WRF on a Multiple Ch Element | |
| 8000 | Channel Lb Rb on a Multiple Channel Element | |
| 8000 | Channel Lb Rb on a Multiple Channel Element | |
| 8000 | Levee on a Multiple Channel Element | |
| 8000 | Multiple Channel Element on a Multiple Channel Element | |
| 8000 | Levee on a Multiple Channel Element | |
| 8000 | Multiple Channel Element on a Multiple Channel Element | |

| |
|---|
| Move the inflow node. |
| Remove the ARF/WRF. |
| Remove the ARF/WRF. |
| Remove the ARF/WRF. |
| A multiple channel cannot be assigned to a bank element.  See reference manual. |
| A multiple channel cannot be assigned to a bank element.  See reference manual. |
| Make sure the multiple channel is on the correct side of the levee. |
| Delete one of the repeated lines in MULT.DAT. |
| Make sure the multiple channel is on the correct side of the levee. |
| A multiple channel cannot be assigned to a bank element.  See reference manual. |