FLO-2D
USERS' MANUAL

Version 2003.06
July 2003

Property of
Flood Control District of MC Library
Please Return to
2801 W. Durango
Phoenix, AZ 85009

FLO-2D
USERS MANUAL

Version 2003.06
July 2003

P.O. Box 66
Nutrioso, AZ 85932

Phone and FAX: (928) 339-1935
Email: flo2d@cybertrails.com

A few comments on modeling free surface flows...

Modeling unconfined free surface flood flows can be complex. Having knowledge of the study area fluvial system, some modeling experience and a set of plausible assumptions will help get you started. With faster computers and high resolution digital terrain models, flood routing models are becoming very detailed. It is necessary for the user to find a balance between model resolution, computer resources and budget. When applying a two-dimensional flood routing model, a number of factors should be considered including the accuracy of the flood hydrology, map resolution, spatial variability of channel cross section data, and limited calibration data. Flood hazard delineation is dependent on hydraulic models and too often we are not critical enough of the model engines, modeling assumptions, limited data bases or the modeling results.

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

As flood modeling advances with hydrograph routing and mapping of unconfined flows, improved graphic tools and more extensive flood data bases will be necessary to support accurate models. If it appears that flood modeling complexity is becoming overwhelming, take heart in the comments of Cunge et al. (1980),

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

As a final word, please remember that software models have glitches and require constant updating. Even when a hydraulic model engine is fine tuned, adding components can introduce conflicts with unrelated subroutines or identify bugs that were previously undetected. FLO-2D is certainly no exception. We will immediately address all questions concerns over model application, accuracy or problems. On occasion, there is a project application that pushes the model to new limits. Such modeling problems can lead to new developments that benefit all users. The modeler is encouraged to share any interesting projects with us. We aspire to make the FLO-2D model a productive and enlightening tool.

FLO-2D SOFTWARE LICENSE AGREEMENT

PLEASE READ CAREFULLY

This end user license agreement ("Agreement") is a legal contract between you (either an individual or a single business entity) ("Licensee") and FLO-2D Software, Inc. By clicking the "I Agree" button below or by installing or otherwise using the software application, you agree to be bound by the terms and conditions of this Agreement. If you do not agree to the terms and conditions of the agreement, do not install or use the FLO-2D Software. The following FLO-2D Software license terms are binding upon any purchaser or Licensee who uses this software.

GRANT OF RIGHTS

FLO-2D Software Inc. grants to Licensee, and Licensee hereby accepts, a non-exclusive, non-transferable, royalty-free license, for use by Licensee only, of the FLO-2D Software package that includes the FLO-2D computer model and processor programs commonly known and referred to as FLO-2D. The license includes the right to copy the FLO-2D Licensed Software only as it is necessary for use within the home or office of purchase and for archival purposes. The license permits the use of the software by Licensee or its regular employees on any and all computers owned by Licensee within or located at the home or office of purchase as indicated by the Licensee's address appears in documents submitted to FLO-2D. The license granted above does not include the right to copy or distribute the FLO-2D Licensed Software outside the home or office of purchase or to any other person. The license does not permit the use of the FLO-2D Licensed Software on a laptop or portable computer outside of the home or office of purchase.

The license granted above does not provide for any free updates of the Licensed Software even though updates may, from time to time, be made available to Licensee. Should such updates be provided to Licensee, this license agreement shall apply to the updated version of the software, unless superseded or supplemented by other license terms provided with the updated version.

The parties agree that all rights, including, but not limited to, rights under the federal copyright laws in and to modifications, if any, to the FLO-2D Licensed Software shall remain the sole and exclusive property of FLO-2D. The parties further agree that all rights, including but not limited to rights under the federal copyright laws in and to the Licensed Software shall remain the sole and exclusive property of FLO-2D Software, Inc. No rights or licenses to the Licensed Software, other than those granted herein are granted, whether expressly, by implication, estoppels or otherwise. The term of this license shall be perpetual. No modification of this License shall be binding on the parties hereto unless such modification is in writing and duly signed by both parties.

Any attempted assignment of this License or any rights or obligations hereunder, without the prior written consent of FLO-2D, shall be null and void and of no effect and a material breach and default of this license agreement.

RESTRICTIONS

The FLO-2D model and its accompanying software and processor programs may not be sold, resold, leased, lent, rented or distributed to any other individual or organization outside the home or office of purchase. The FLO-2D model and accompanying software and processor programs can not be copied outside the home or office of purchase or to any other person, decompiled, disassembled, reverse engineered, recreated as a derivative program or otherwise used except as stated in this agreement.

DISCLAIMER OF WARRANTIES AND LIMITATIONS OF LIABILITY

FLO-2D Software, Inc. does not make any warranty, either express or implied with respect to the licensed software, its quality, merchantability, or fitness for a particular purpose. All the FLO-2D Licensed Software provided hereunder is licensed "AS IS" and does not warrant that the licensed software is free from claims of infringement or patents, copyrights, trade secrets, or other proprietary rights of others. There are no warranties, either express or implied, and any and all such warranties are hereby disclaimed and negated. FLO-2D Software and its employees do not warrant the performance or results that you may obtain by using the FLO-2D Software or any results generated by the software. The user assumes the entire risk of using the FLO-2D Software. No oral or written information or advice given by FLO-2D Software Inc. or its employees shall create a warranty or make any modification, extension or addition to this warranty. In no event whatsoever, shall FLO-2D Software, Inc. or its employees be liable to the Licensee or to any third parties for any damages caused, in whole or in part, by the use of the licensed software or for any lost revenues, damages to computers or other computer software, lost profits, lost savings or other direct or indirect, incidental, special, or consequential damages incurred by any person, even if advised of the possibility of such damages or claims, arising out the use or application of the FLO-2D Software or the inability to use the software.

The liability of FLO-2D Software, Inc. for a defective copy of the FLO-2D Software will be limited exclusively to the replacement of the originally purchased copy of the FLO-2D Software with another copy of the software or the refund of the initial license fee if the originally purchased copy is returned within 30 days of the date of purchase.

DEFAULT

Without prejudice to any other rights, FLO-2D Software, Inc shall have the right to terminate this Agreement and the license granted herein if the Licensee fails to comply with or commits a material breach of the terms and conditions of this Agreement or commits an act of or is subject to a Default. A "Default" means any one or more of the following events: The distribution, exchange, or offer or promise to distribute or exchange one or more copies of the FLO-2D Software by the Licensee, whether by sale, license, lease or otherwise, and whether or not any consideration is received for any such transfer or offer or promise. Upon the occurrence of a Default, FLO-2D shall provide written notice of the Licensee and the Licensee shall have fifteen (15) days from the Licensee's receipt of said notice of Default to cure the same. If the Licensee does not affect such a cure within the prescribed time, then this Agreement and the FLO-2D license shall be terminated. Within fifteen (15) calendar days after the Licensee's receipt of notice as provided for above, LICENSEE shall deliver to FLO-2D Software, Inc. all copies, including but not limited to, all archival and backup copies for the FLO-2D Software and all documentation related thereto.

GOVERNING LAW

This License shall be deemed made and accepted in and governed by the laws of the State of Arizona. The state and federal courts located in Arizona shall have non-exclusive jurisdiction and venue to hear all disputes arising out of or related to this License.

COPYRIGHT

© Copyright 1989, 1993. FLO-2D is copyrighted by J. S. O'Brien. All rights reserved. The FLO-2D software and manual are protected by U.S. Copyright Law (Title 17 US Code). Unauthorized reproduction and/or sales may result in imprisonment and/or fines (17 USC 506). Copyright infringers may also be subject to civil liability.

BRIEF OVERVIEW

FLO-2D is a simple volume conservation model that distributes a flood hydrograph over a system of square grid element (tiles). It is a two-dimensional flood routing model that can be valuable tool for delineating flood hazards, regulating floodplain zoning or designing flood mitigation. FLO-2D numerically routes a flood hydrograph while predicting the area of inundation and simulating floodwave attenuation. The model is effective for analyzing river overbank flows, but it can also be used to analyze unconventional flooding problems such as unconfined flows over complex alluvial fan topography and roughness, split channel flows, mud/debris flows and urban flooding. Conventional one-dimensional, single discharge flood analysis can be replaced with a detailed FLO-2D model that includes rainfall and infiltration, levees, hydraulic structures, streets, hyperconcentrated sediment flows and the effects of buildings or flow obstructions.

FLO-2D simulates unconfined overland flow using topographic data files that have been developed from a digital terrain model or digitalized base map. The FLO-2D software package includes a grid developer system (GDS) that will overlay a square grid system on a set of random digital terrain (DTM) points. The GDS will filter DTM points, interpolate the DTM data and assign elevations to grid elements. As an option, topographic digital maps prepared with a computer-aided design and drafting (CADD) program can also be assigned a grid system.

The user has control over the creation of spatial and temporal output data files. The FLO-2D results including maximum flow depth and velocity can be viewed graphically in the MAPPER post processor program. MAPPER will generate very detailed flood inundation color contour mapping by subtracting the DTM points from the predicted FLO-2D water surface elevations to generate an array of flow depths associated with every DTM point. The MAPPER program automates flood hazard delineation. The results can also be re-imported to the original mapping using CADD software to produce maximum depth and velocity contours.

The FLO-2D manual is divided into two sections. The first half of the manual is devoted to a model description, theory and components. The users is encouraged to read this section to become familiar with the overall model attributes and equations. The second half of the manual is subdivided into a series of data files with variable descriptions and comments. The user should consult this portion of the manual when constructing data files. These data files can be viewed in any ASCII text editor. A graphical user interface (GUI) has been developed to assist the user in preparing and editing the data files.

FLO-2D is on FEMA's list of approved hydraulic models for both riverine and unconfined alluvial fan flood studies. It has been extensively by a number of federal agencies including the Corps of Engineers, Bureau of Reclamation, USGS, NRCS, Fish and Wildlife Service and the National Park Service. FLO-2D has been used on hundreds of projects by consultants worldwide. The user can keep current on FLO-2D model and processor updates, short courses and other modeling news at the website: www.flo2d.com.

TABLE OF CONTENTS

Page

List of Figures vii
List of Tables viii

I. INTRODUCTION 1

1.1 Evolution of the FLO-2D Model 1
1.2 Modeling the Hydrologic System with FLO-2D 2
1.3 Getting Started on a Project 6

II. FLO-2D MODEL THEORY 8

2.1 Governing Equations 8
2.2 FLO-2D Logic How the Model Works 10
2.3 The Importance of Volume Conservation 15

III. MODEL COMPONENTS 16

3.1 Model Features and Grid System 16
3.2 Overland Flow 17
3.3 Channel Flow 22
3.4 Floodway Routine 26
3.5 Channel-Floodplain Interface 27
3.6 Limiting Froude Numbers 27
3.7 Levees 28
3.8 Hydraulic Structures 29
3.9 Street Flow 30
3.10 Floodplain Surface Storage Area Modification 31
3.11 Rainfall 33
3.12 Infiltration and Abstraction 33
3.13 Evaporation 36
3.14 Overland Multiple Channel Flow 36
3.15 Sediment Transport - Total Load 37
3.16 Mud and Debris Flow Simulation 40

IV. MODEL CONCEPTUALIZATION 51

4.1 Limitations and Assumptions 51
4.2 Finite Difference Routing Algorithm 52
4.3 Parameter Sensitivity 55
4.4 Floodplain and Channel Inflow and Outflow Hydrographs 57

TABLE OF CONTENTS (cont.)

Page

4.5 Floodplain Cross Sections 57
4.6 Graphical User Interface and Working Environment 58
4.7 Grid Developer System (GDS) 58
4.8 Graphical Output Displays 60
4.9 Data Output Options 61
4.10 Starting the Program 66
4.11 Code Language and Hardware Requirements 67

V. FLO-2D APPLICATIONS AND METHODS 68

5.1 River Applications 68
5.2 Overland Flow and Alluvial Fan Applications 69
5.3 Project Results - What is a Successful Flood Simulation 70

VI. FLO-2D MODEL VERIFICATION 72

6.1 General 72
6.2 Verification Test 1. Channel Hydraulics in the California Aqueduct 73
6.3 Verification Test 2. Variable Channel Geometry Hydraulics 75
6.4 Verification Test 3. River Floodplain Simulation - Truckee River, Reno, Nevada 77
6.5 Verification Test 4. River Flood Routing - Green River, Utah 82
6.6 Verification Test 5. Floodplain Inundation - Middle Rio Grande, New Mexico 86
6.7 Verification Test 6. Verification of Mudflow Hydraulics 88

VII. FLO-2D PROJECT APPLICATIONS 90

7.1 Flood Hazard Delineation, Mapping and Results 90
7.2 Partial List of FLO-2D Projects 90

VIII. REFERENCES 98

LIST OF FIGURES

Page

Figure 1. Physical Processes Simulated by FLO-2D 4
Figure 2. Channel - Floodplain Interface 5
Figure 3. FLO-2D Flow Chart 7
Figure 4. Discharge Flux Across Grid Element Boundaries 13
Figure 5. FLO-2D Stability Criteria Flow Chart 14
Figure 6. Conceptual Octagonal for the Boundary Widths and Flow Lengths 17
Figure 7. Overland Flow Subroutine Flow Chart 21
Figure 8. Channel Extension over Several Grid Elements 22
Figure 9. Levees Depicted in Red and Rivers in Blue in the FLOENVIR Program 29
Figure 10. Streets Depicted in Green in the FLOENVIR Program 31
Figure 11. Area and Width Reduction Factors 32
Figure 12. Shear Stress as a Function of Shear Rate for Fluid Deformation Models 43
Figure 13. Dynamic Viscosity of Mudflow Samples Versus Volumetric Concentration 47
Figure 14. Yield Stress of Mudflow Samples Versus Volumetric Concentration 48
Figure 15. Classification of Hyperconcentrated Sediment Flows 48
Figure 16. Channel Profile 76
Figure 17. Comparison of FLO-2D and HEC-2 Flow Depths for 5,000 cfs Discharge 76
Figure 18. Comparison of FLO-2D and HEC-2 Flow Velocities for 5,000 cfs Discharge ... 77
Figure 19. Predicted Truckee River Water Surface Profiles for HEC-2 and FLO-2D 79
Figure 20. Truckee River Observed and FLO-2D Predicted Area of Flood Inundation 80
Figure 21. FLO-2D Predicted Stage for the 1997 Flood Event at the Reno Gage 81
Figure 22. FLO-2D Predicted Stage for the 1997 Flood Event at the New Vista Gage 81
Figure 23. 1997 High Flow Season Hydrograph at Jensen, Utah 83
Figure 24. November 1998 Power Plant Discharge Release from Flaming Gorge Dam 84
Figure 25. FLO-2D Predicted vs. Measured Discharge, Jensen USGS For Power Plant Releases at Flaming Gorge Dam 85
Figure 26. Area of Inundation, Middle Rio Grande, May, 1992 87
Figure 27. Rudd Creek Mudflow Maximum Flow Depth Contours 89

LIST OF TABLES

Page

Table 1. Guidelines for Selecting a Flood Routing Method 9
Table 2. Overland Flow Manning's n Roughness Values 18
Table 3. Initial Abstraction 34
Table 4. Green-Ampt Infiltration Parameters - Hydraulic Conductivity and Porosity 35
Table 5. Green-Ampt Infiltration Parameters - Soil Suction 35
Table 6. Green-Ampt Infiltration Parameters - Volumetric Moisture Deficiency 36
Table 7. Resistance Parameters for Laminar Flow 44
Table 8. Yield Stress and Viscosity as a Function of Sediment Concentration 46
Table 9. Mudflow Behavior as a Function of Sediment Concentration 49
Table 10. California Aqueduct Predicted Hydraulics 74

Two Dimensional Flood Routing Model

I. INTRODUCTION

The first portion of the manual is a reference document that describes the physical processes of flooding. It is designed to acquaint the user with the model theory, finite difference algorithms, model components, modeling assumptions and limitations, and potential flood scenarios. A reference list is also provided for further reading. The second half of the manual provides a detailed description of the required input and output data files. A series of processor programs is provided to assist data file development and to graphically view results. Some example project applications are provided for the user to test and modify. These examples can be used as templates for developing your own project. The website: www.flo-2d.com has additional information on model updates and other example projects.

1.1 Evolution of the FLO-2D Model

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

The original MUDFLOW model was only a few hundred lines of Fortran code and was limited to 250 grid elements. A six hour hydrograph took over 12 hours to run on an XT computer. With the advent of faster pc's, additional components were added and the numerical routing algorithm was improved. After 15 years of development, the program code has grown to be in excess of 25,000 lines, 60 subroutines and a number of processor programs. It has been integrated into a Windows format. Virtually none of the original simplistic DHM concept remains in the current FLO-2D model. FLO-2D computes overland flow in 8-directions, reports on mass conservation, utilizes a new timestep incrementing and decrementing scheme, incorporates efficient numerical stability criteria, has unlimited array allocation (unlimited overland and channel grid elements) and includes graphical editing and output display processor programs.

FLO-2D has grown into a physical process model that routes rainfall-runoff and flood hydrographs over unconfined flow surfaces or in channels using either a diffusive or dynamic wave approximation to the momentum equation. The model now has a number of components to simulate street flow, buildings and obstructions, sediment transport and mobile bed, spatially variable rainfall and infiltration, floodways and many other flooding details. Predicted flow depth and velocity between the grid elements represent average hydraulic flow conditions computed for a small timestep (on the order of seconds). Typical applications have grid elements that range from 25 ft to 500 ft on a side and while number of grid elements is unlimited, the user is cautioned to avoid extremely large models. One model of over 400,000 grid elements took several days to simulate a three hour hydrograph flood.

1.2 Modeling the Hydrologic System with FLO-2D

To model the hydrologic system, FLO-2D consists of a series of components and processor programs that break up a flood simulation into a number of discretized small units. The Grid Developer System (GDS) generates a grid system that represents the topography as a series of small tiles. The FLO-2D flood model has components for rainfall, channel flow, overland flow, street flow, infiltration, levees and other physical features. The GDS and the FLOENVIR processor programs are used to spatially edit the grid system attributes. PROFILES is a processor program to edit channel slope and shape. Flood routing results can be viewed graphically in the MAXPLOT, MAPPER and HYDROG (view hydrographs) programs.

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

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

The user is encouraged to utilize all these components while understanding the importance of each component to the overall flood distribution. It is important to assess the level of detail required on a given project. FLO-2D users have a tendency to put more detail into their models than is necessary. The modeling detail and accuracy predicted water surface elevations should be consistent with the resolution of the mapping, survey data and the hydrologic data base. Simulating large flood events requires less detail than shallow flood models or the design of flood mitigation measures. Selection of the grid element size for most flood inundation projects range from 25 ft to 500 ft. Different scales of resolution can be applied on the same project. For example, a 15 mile reach of river can be modeled using a 500 ft grid system. A short one mile reach where levees are proposed can be modeled with a 50 ft grid system to investigate the potential for levee failure. The hydrographic output from the 15 mile reach will constitute the input to the one mile river reach. Preparation of channel flow, street flow, buildings and flow obstructions data files can be time consuming and should be tailored to meet the project needs.

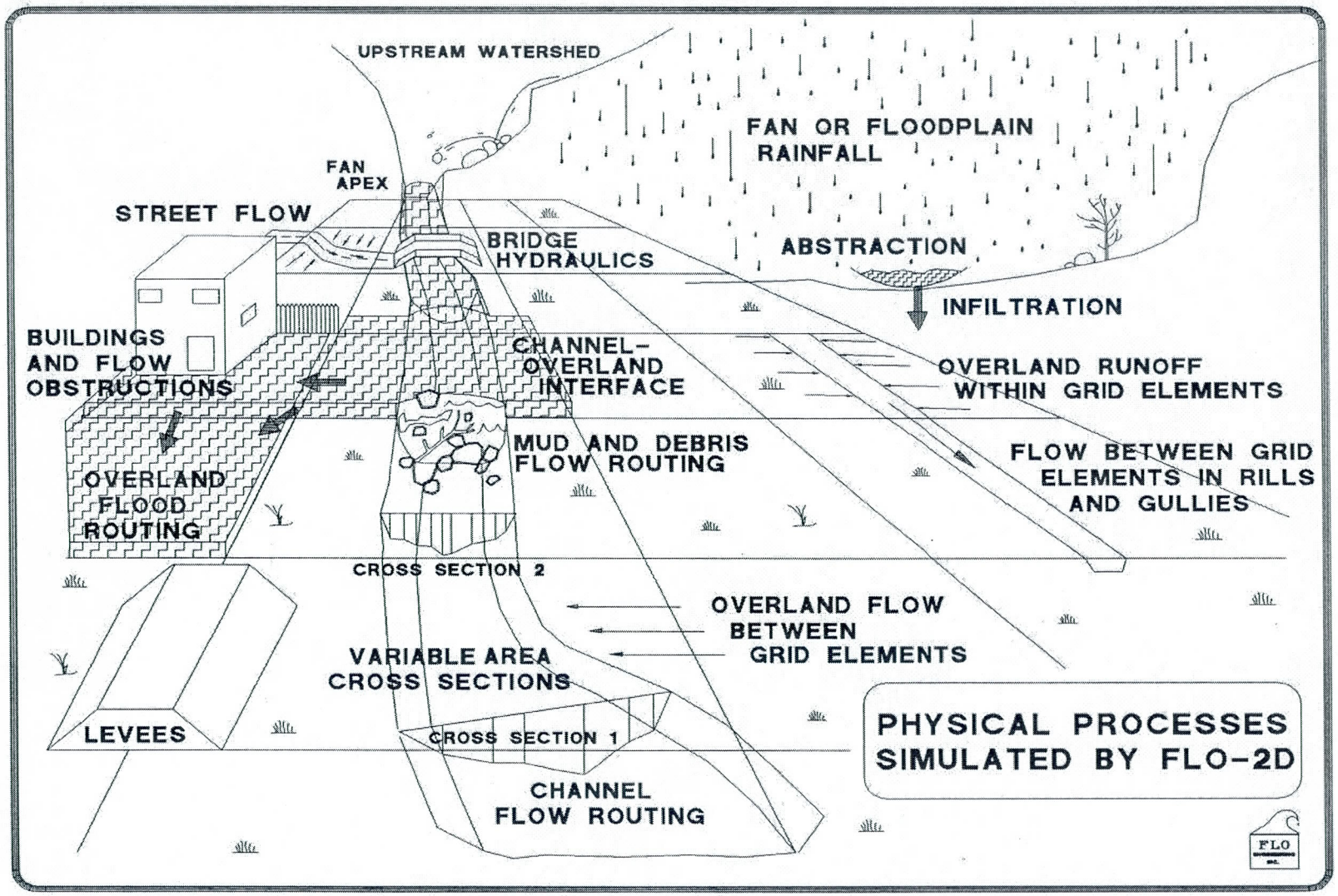


Figure 1. Physical Processes Simulated by FLO-2D

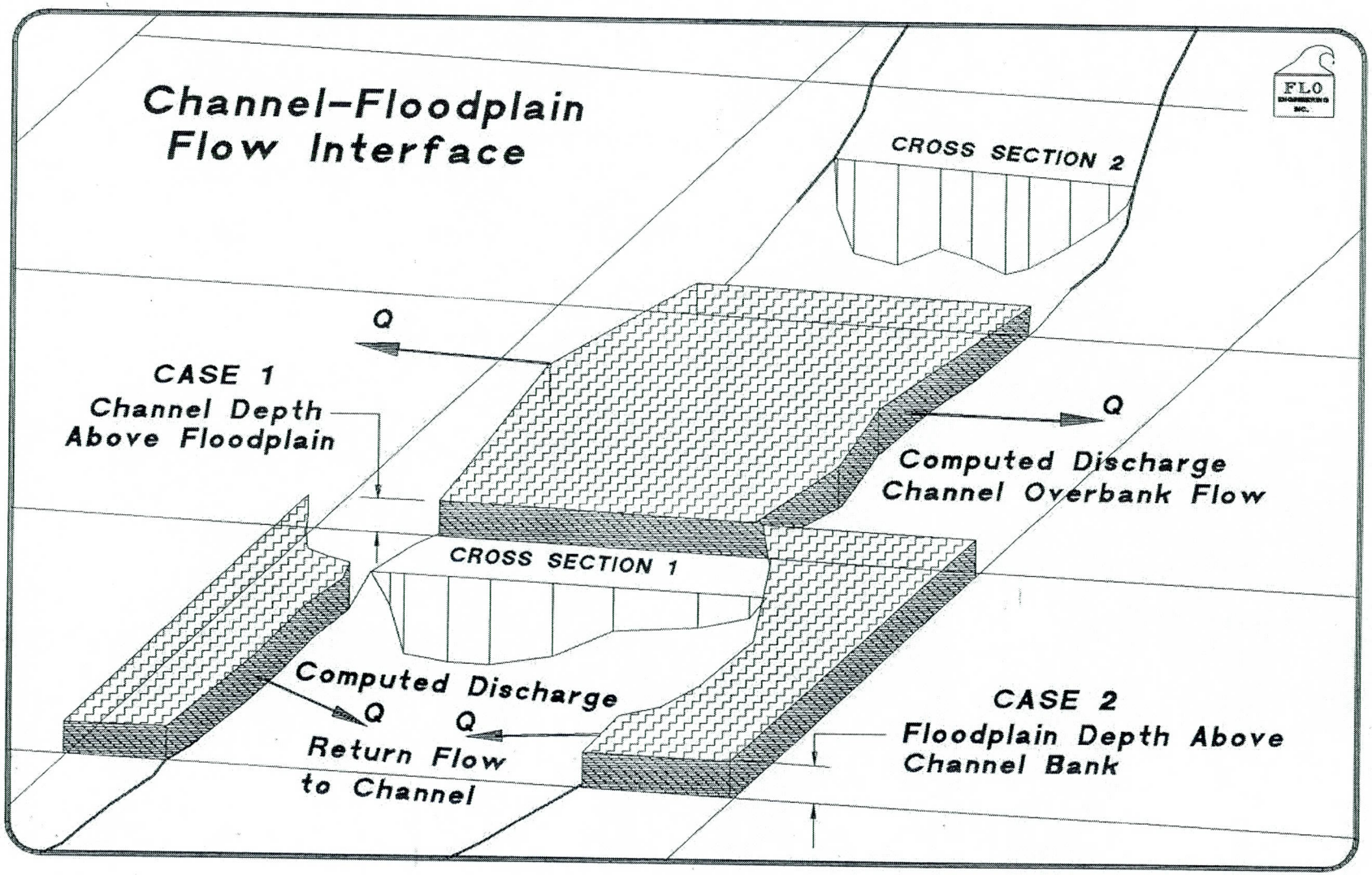


Figure 2. Channel – Floodplain Interface

1.3 Getting Started on a Project

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

The second step arises from the fact that each flood simulation requires an inflow flood hydrograph or a rain storm. The discharge inflow points might include the alluvial fan apex or a known discharge location in a river system. FLO-2D can be used to generate the flood hydrograph at a specific location using a rainfall-runoff simulation in the upstream watershed. Another approach is to use hydrologic program such as the Corps' HEC-1 model to generate an inflow hydrograph for the FLO-2D model. Rainfall can be then simulated on the flood inundation surface as the flood processes over the grid system. The model inflow flood volume is the primary factor that determines an area of flood inundation. For that reason, it is suggested that a comparable effort be spent on the hydrologic investigation as is being expended on the flood routing analysis.

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

In the following sections of this User Manual, modeling theory and a description of the FLO-2D components is presented. The user should refer to these sections when developing the individual data files. Data tables are presented in the following sections that will guide the user in selecting various component parameters such as roughness values or hydraulic conductivity for infiltration.

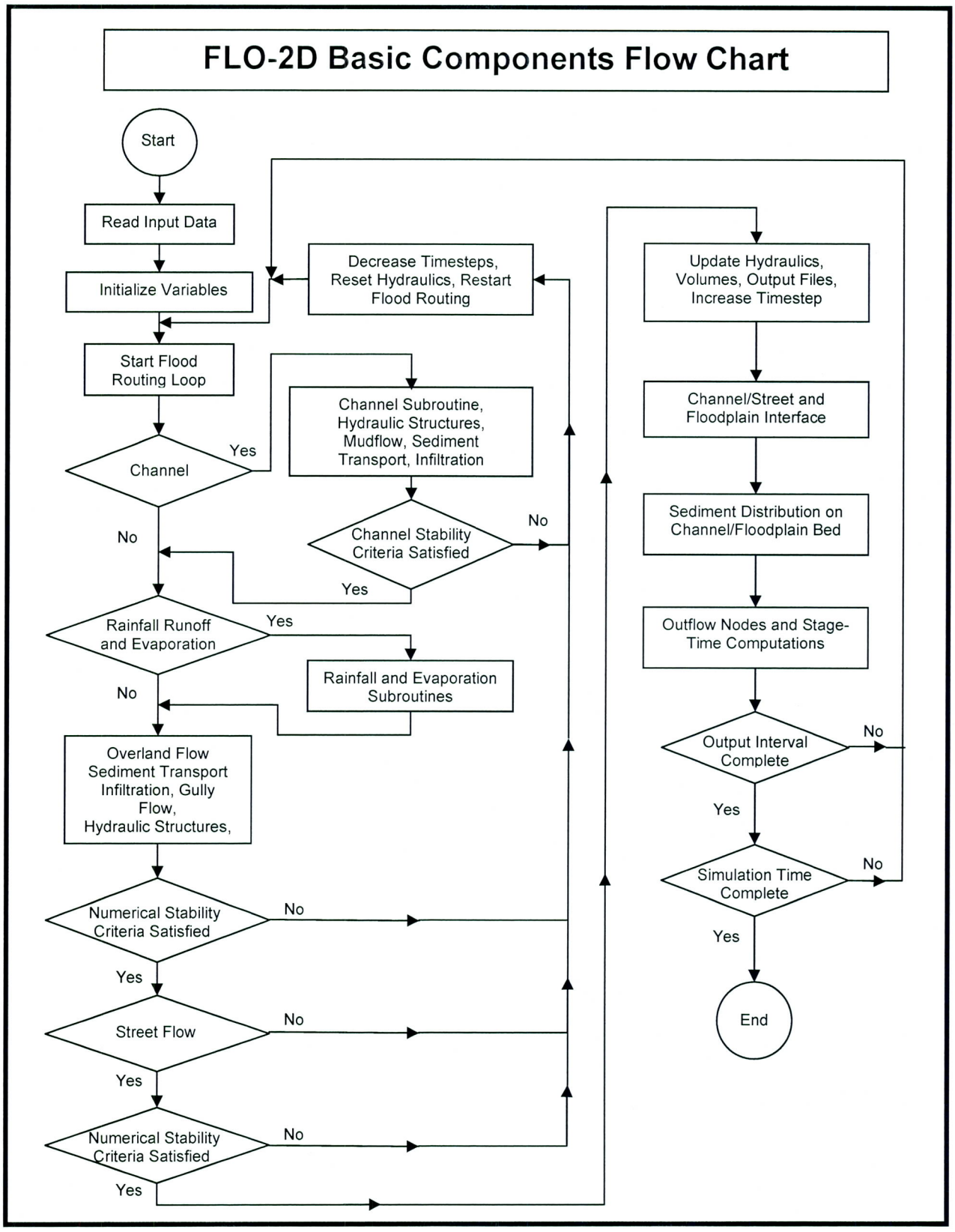


Figure 3. FLO-2D Flow Chart

II. FLO-2D MODEL THEORY

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

2.1 Governing Equations

$$\frac{\partial h}{\partial t} + \frac{\partial h V_x}{\partial x} + \frac{\partial h V_y}{\partial y} = i$$

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

$$S_{fx} = S_{ox} - \frac{\partial h}{\partial x} - \frac{V_x}{g}\frac{\partial V_x}{\partial x} - \frac{V_y}{g}\frac{\partial V_x}{\partial y} - \frac{1}{g}\frac{\partial V_x}{\partial t}$$

$$S_{fy} = S_{oy} - \frac{\partial h}{\partial y} - \frac{V_y}{g}\frac{\partial V_y}{\partial y} - \frac{V_x}{g}\frac{\partial V_y}{\partial x} - \frac{1}{g}\frac{\partial V_y}{\partial t}$$

where h is the flow depth and V_x and V_y are the depth-averaged velocity components along the x- and y-coordinates. The excess rainfall intensity (i) may be nonzero on the flow surface. The friction slope components S_{fx} and S_{fy} are written as function of bed slope S_{ox} and S_{oy}, pressure gradient and convective and local acceleration terms. A diffusive wave approximation to the equations of motion is defined by neglecting the last three acceleration terms. By neglecting the pressure gradient term, a kinematic wave representation of the momentum equation is derived. For a discussion of the kinematic wave application in numerical modeling see Ponce et al., 1978. The kinematic wave equation is not used in the FLO-2D model. The FLO-2D user can select between the diffusive wave and the full dynamic wave versions of the momentum equation.

The two-dimensional representation of the equations of motion in FLO-2D is better defined a quasi two-dimensional model using a square finite difference grid system. The equation of motion is solved by computing the average flow velocity across a grid element boundary one direction at time. There are eight potential flow directions, the four compass directions (north, east, south and west) and the four diagonal directions (northeast, southeast, southwest and northwest). Each velocity computation is essentially one-dimensional in nature and is solved independently of the other seven directions. The individual pressure, friction, convective and local acceleration components in the momentum equation are retained. More discussion of model solution and constitutive equations will be presented in the next section.

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

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

This comparison of terms illustrates that on most steep slopes, the application of the kinematic wave is sufficient to model floodwave progression and the contribution of the acceleration terms can be neglected. The addition of the pressure gradient term to create the diffusive equation will enhance overland flow simulation with complex topography. The diffusive wave equation with the pressure gradient is required if the grid system has topographic depressions. Explicit numerical schemes to solve the diffusive wave equation generally require relatively mild slopes. Flat slopes may induce very small timesteps to achieve numerical stability. The local and convective acceleration terms are important to the solution for channel flow especially for flat or adverse slopes or very steep slopes.

It is recommended that the full dynamic wave equation be applied in the FLO-2D model for applications except a simple overland flow simulation on a mild or steep slope. In this case, very little accuracy is sacrificed when the diffusive wave equation is used compared to the full dynamic model (Akan and Yen, 1981). Criteria for selecting a channel routing equation is given in the following table (adapted from the Corps of Engineers, Technical Engineering and Design Guidelines No. 19, 1997).

Table 1. Guidelines for Selecting a Flood Routing Method

Physical Processes[1]	Recommended Routing Equation
1. Steep alluvial fans, watersheds or floodplains	Diffusive wave, kinematic wave
2. Mild slope floodplains, backwater areas which influence hydrograph, overbank flows	Dynamic wave, diffusive wave
3. Bed slope > 10 ft/mile and TS v/d > 171	Dynamic wave, diffusive wave, kinematic wave
4. Bed slopes ~ 2 to 10 ft/mile and TSv/d < 171	Dynamic wave, diffusive wave
5. Bed slope < 2 ft/mile and $TS (g/d)^{0.5} > 30$	Dynamic wave, diffusive wave
6. Bed slope < 2 ft/mile and $TS (g/d)^{0.5} < 30$	Dynamic wave

[1]T = hydrograph peak time of rise; S = bed slope; v = average flow velocity; d = average flow depth; g = gravitational acceleration (32.2 ft/s^2)

2.2 FLO-2D Logic - How the Model Works

The differential form of the continuity and momentum equations in the FLO-2D model is solved with a central, finite difference scheme. This explicit algorithm solves the momentum equation for the flow velocity across the grid element boundary one element at a time. Explicit numerical schemes are simple to formulate but usually are limited to small timesteps by strict numerical stability criteria. Finite difference explicit numerical schemes require significant computational time when simulating complex flows hydraulics such as slowly rising flood waves, channels with non-prismatic features, abrupt changes in slope, tributaries or split flow and ponded flow areas.

The solution domain is discretized into uniform, square grid elements. The computational procedure for overland flows involves calculating the discharge across each of the boundaries in the eight potential flow directions. The flow directions include the four compass directions and the four diagonal directions (Figure 4). Each grid element hydraulic computation begins with an estimate of the linear flow depth at the grid element boundary. The estimated boundary flow depth is an average of the flow depths in the two grid elements that will be sharing discharge in one of the eight directions. Although a number of non-linear estimates of the boundary depth were attempted in earlier versions of the model, they did not significantly enhance or improve the results. The other hydraulic parameters are also averaged to compute the flow velocity including flow resistance (Manning's n-value), flow area, slope, water surface elevation and wetted perimeter.

The floodplain flow velocity at the boundary is the dependent variable. FLO-2D will solve either the diffusive wave equation or the full dynamic wave equation to compute the velocity. Manning's equation is then applied in one direction using the average difference in the water surface slope to compute the velocity. If the diffusive wave equation is selected, the velocity is then computed for all eight potential flow directions for each grid element. If the full dynamic wave momentum equation option is applied, the computed diffusive wave velocity is used as the first approximation (the seed velocity) in the Newton-Raphson second order method of tangents for determining the roots of the full dynamic wave equation which is a second order, non-linear, partial differential equation. The local acceleration term is the difference in the velocity for the given flow direction over the previous timestep. The convective acceleration term is evaluated as the difference in the flow velocity across the grid element from the previous timestep. The local acceleration term ($1/g*\partial V/\partial t$) for grid element 251 in the east (2) direction converts to:

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

where V_t is the velocity in the east direction for grid element 251 at time t, V_{t-1} is the velocity at the previous timestep (t-1) in the east direction, Δt is the timestep in seconds, and g is the acceleration due to gravity. A similar construct for the convective acceleration term can be made.

The discharge across the grid element boundary is computed by multiplying the velocity times the cross sectional flow area. After the discharge is computed for all eight directions, the net change in discharge (sum of the discharge in the eight flow directions) in or out of the grid element is multiplied by the timestep to determine the net change in the water volume (see Figure 4). This net change in volume is then divided by the available surface area (storage area) on the grid element to obtain the increase or decrease in flow depth for the timestep. The channel routing integration is performed essentially the same way except that the flow depth is a function of the channel cross section geometry and there are usually only one upstream and one downstream channel grid element for sharing discharge. The channel routing algorithm will be explained in more detail in a later section.

The key to efficient finite difference flood routing is the numerical stability criteria that limit the magnitude of the timestep. FLO-2D has a variable timestep that varies depending on whether the numerical stability criteria are not exceeded or not. The numerical stability criteria are checked for the every grid element on every timestep to ensure that the model solution algorithms converge and that the solution is stable. If the numerical stability criteria are exceeded, the timestep is decreased and all the previous hydraulic computations for that timestep are discarded. The FLO-2D flood routing scheme proceeds on the basis that the timestep is sufficiently small to insure numerical stability. Most explicit schemes are subject to the Courant-Friedrich-Lewy (CFL) condition for numerical stability (Jin and Fread, 1997) that related the floodwave celerity to the model time and spatial increments. FLO-2D uses the CFL condition for the floodplain, channel and street routing. The timestep Δt is limited by:

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

where:

C is the Courant number ($C < 1.0$)
Δx is the square grid element width
v is the computed average cross section velocity
c is the computed wave celerity

While the coefficient C can vary from 0.3 to 1.0 depending on the type of explicit routing algorithm, a value of 1.0 is employed in the FLO-2D model to allow the model to have the largest timestep under this criteria.

IMPORTANT NOTE: The CFL stability criteria is hardwired in the model, the user does not input any data for this stability analysis.

For full dynamic wave routing, another set of the numerical stability criteria is applied that was developed by Ponce and Theurer (1982). This criteria is a function of bed slope, specific discharge and grid element size. It is expressed as:

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

where q_o is the unit discharge, S_o is the bed slope and ζ is an empirical coefficient (Ponce and Theurer, 1982). The coefficient ζ was created as a variable unique to the grid element and is

adjusted by the model during runtime within a minimum and maximum range set by the user. Similar to the CFS criteria, when this numerical stability is exceeded the hydraulic computations for that timestep are dumped and the timestep is decreased.

Before the CFL and the full dynamic wave equation numerical stability criteria are evaluated in the FLO-2D simulation, the percent change in depth from the previous timestep for a given grid element is checked. This percent change in depth is used to preclude the need for any additional numerical stability analysis. If the percent change in depth is greater than that specified by the user, the timestep is decreased and all the hydraulic computations for that timestep are voided. If numerical stability using these three criteria is not sustained, the volume conservation over several output intervals will be violated and fluid volume will be either generated or lost by the model (see Figure 5).

IMPORTANT NOTE: A 30% percent in depth for the DEPTHTOL variable in the TOLER.DAT file is recommended for most applications. A range of 20% to 30% is suggested for DEPTHTOL.

Timesteps generally range from 0.1 second to 60 seconds. The model starts with the minimum timestep and increases it until one of the three numerical stability condition is exceeded, then the timestep is decreased. If the stability criteria continue to be exceeded, the timestep is decreased until the minimum timestep is reached. If the minimum timestep is not small enough to conserve volume or maintain numerical stability, then the minimum timestep can be reduced, the numerical stability coefficients can be adjusted or the input data can be modified. The timesteps are a function of the discharge flux for a given grid element and its size. Small grid elements with a steep rising hydrograph and large peak discharge require small timesteps. Accuracy is not compromised if small timesteps are used, but the computational time can be very long if the grid system is large.

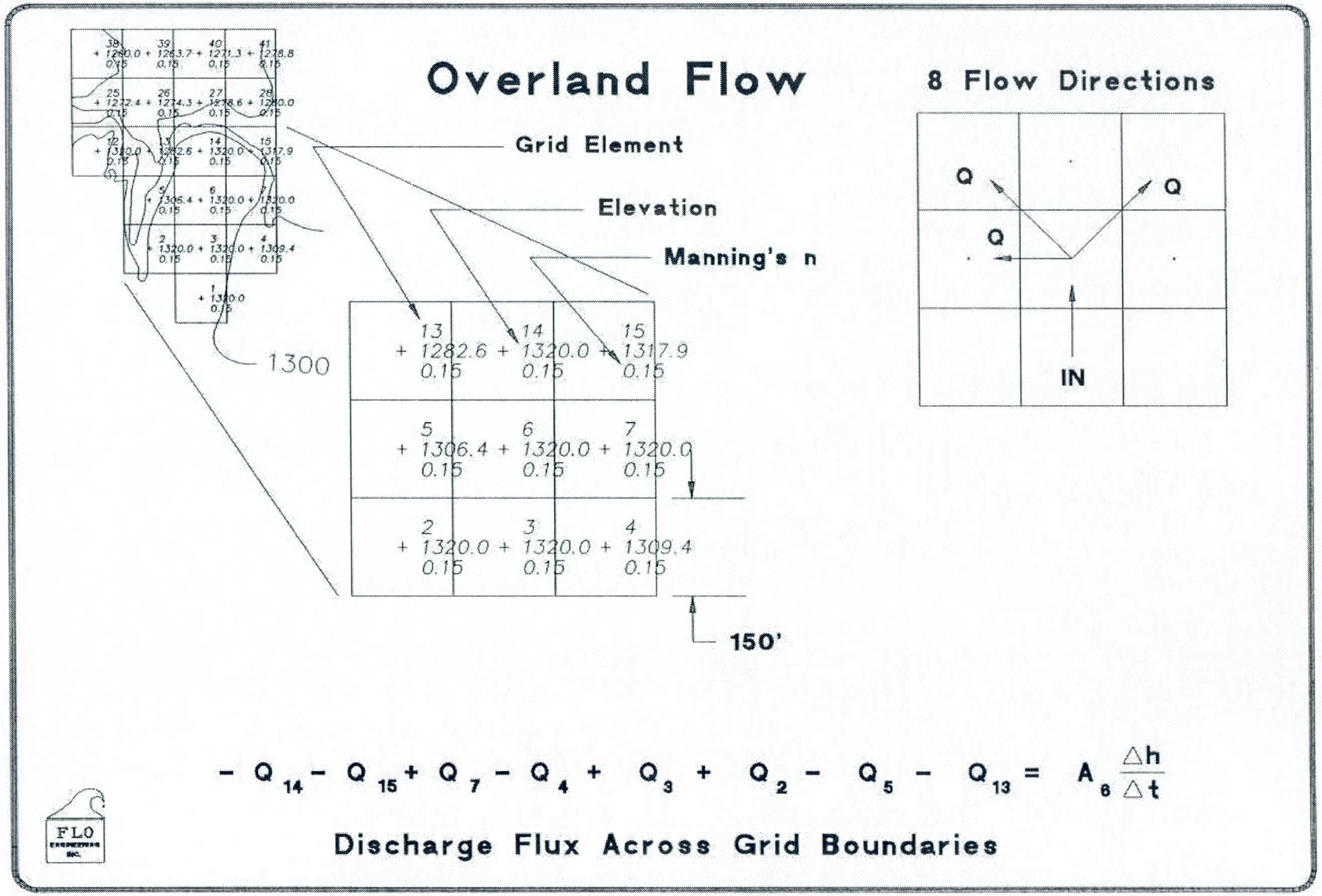


Figure 4. Discharge Flux Across Grid Element Boundaries

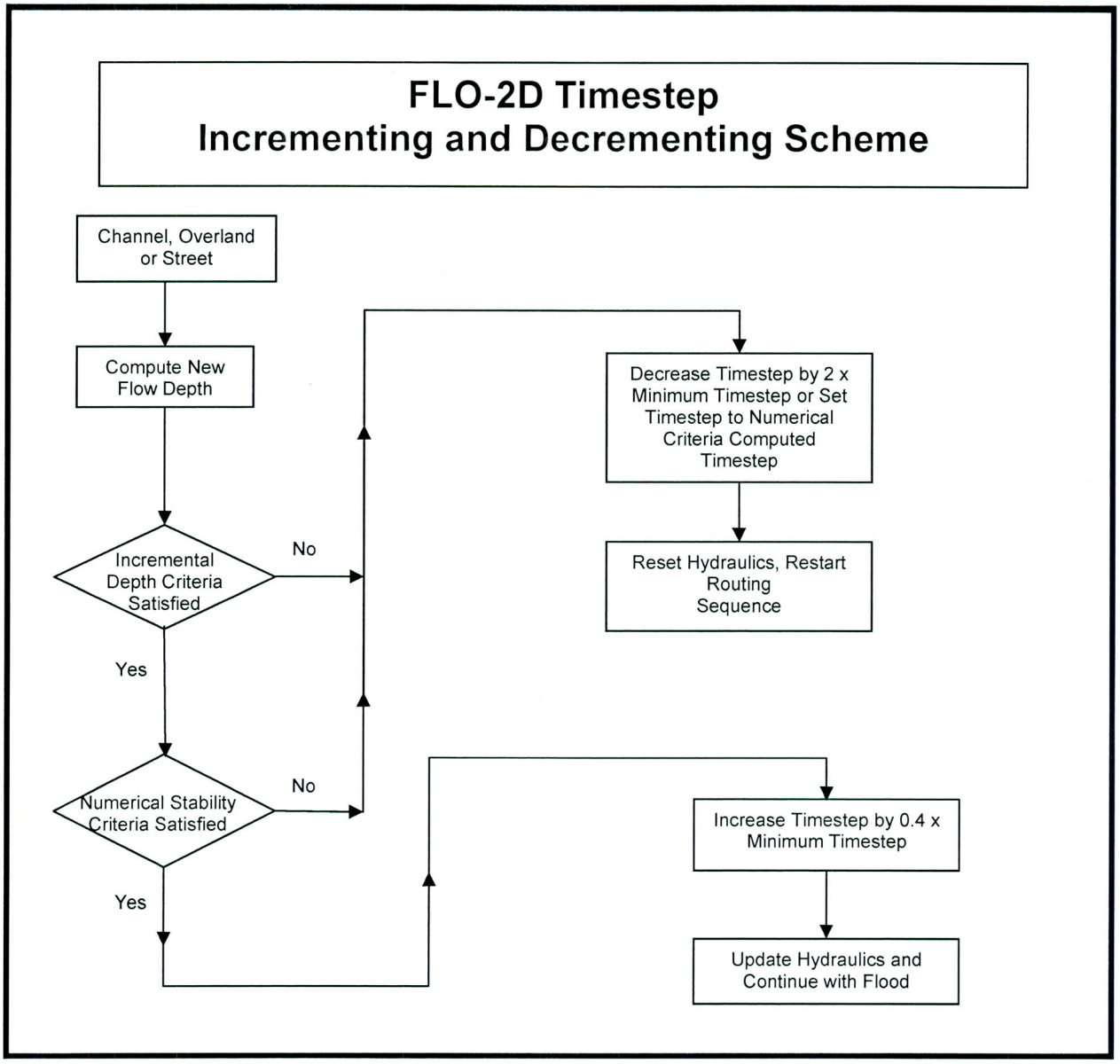


Figure 5. FLO-2D Stability Criteria Flow Chart

2.3 The Importance of Volume Conservation

A review of model flood simulation results begins with volume conservation. Volume conservation indicates flood simulation numerical accuracy and confirms that the millions of calculations during a flood simulation are being consistently performed. The inflow volume, outflow volume, change in storage and infiltration and evaporation losses from the grid system are summed at the end of each time step. The difference between the total inflow volume and the outflow volume plus the storage and losses is the volume conservation. Volume conservation results are written to the output files or to the screen at user specified output time intervals. Data errors, numerical instability, inappropriate or inconsistent simulation techniques will cause a loss of volume conservation. For example, forcing subcritical or supercritical flow with poorly assigned n-values could effect the volume conservation. Volume conservation can be used to debug a model and discern whether model dysfunction occurred in the channel or floodplain components. Any simulation not conserving volume should be revised. It should be noted that volume conservation in any flood simulation is not exact. The user must decide on an acceptable level of error in the volume conservation, generally 0.001 percent or less.

IMPORTANT NOTE: Review the SUMMARY.OUT file to determine how well the model conserves volume for a flood simulation. If a simulation does not conserve volume, the user should determine the location of the data problem by turning off the various components and running the model with only one component (such as streets or channels) turned on at a time.

III. MODEL COMPONENTS

3.1 Model Features and Grid System

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

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

The first step in undertaking a FLO-2D flood simulation is to define the potential flow surface. A grid system can be created with either the grid developer GDS or a CADD program. The product of the GDS is a topographic file (FPLAIN.DAT) that identifies the contiguous grid elements and contains the floodplain roughness and grid element elevation. The major portion of the model data requirements are presented in this automated and error free file. The procedures for creating the grid system and the FPLAIN.DAT file as well as all the data files are discussed in detail in the second portion of the manual.

Data file preparation and computer run times vary according to the number and size of the grid elements, the inflow discharge flux and the duration of the inflow flood hydrograph being simulated. Please keep in mind that the flood events being modeled are generally large enough that the grid elements do not have to be excessively small. Most flood simulations can be accurately performed with grid elements 100 ft to 500 ft on a side. Projects have been undertaken with grid elements as small as 10 ft, although models with grid elements this small are exceedingly slow. One project simulated flow in a street intersection and the grid system was generated inside the streets. Grid system size selection and data acquisition should

accommodate the project needs. It is important to balance the project detail and the number of model components applied with the mapping resolution and anticipated level of accuracy in the results. It is often more valuable from a project perspective to have a model that runs quickly enabling many simulations to be performed from which the user can learn about how the project responds to flood scenarios rather than invest a major effort into creating a huge grid system that takes days to run. The selection of the grid element size is discussed in detailed in Section 2.6 of the Data Input portion of the manual. Model component selection should focus on those physical features that will significantly effect volume distribution. A brief description of the FLO-2D components follows.

3.2 Overland Flow

The simplest type of FLO-2D model is overland flow on an alluvial fan or floodplain. It requires only the topography files and a hydrograph along with the two control files CONT.DAT and TOELR.DAT. Overland flow is routed in eight possible flow directions (the four compass directions and the four diagonal directions). In this manner, FLO-2D treats each grid element as an octagon rather than a square (Figure 6). The conceptualized grid element octagonal geometry is important primarily to the flow width across the grid element boundaries and diagonal flow lengths. The grid element surface area is still a square.

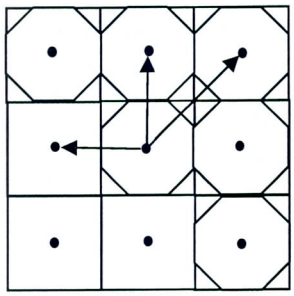


Figure 6. Conceptual Octagon for the Boundary Widths and Flow Lengths

The FPLAIN.DAT file defines the potential flow surface and grid element linkages. It contains the data that identify the grid elements and their neighbors, hydraulic roughness and elevations. The horizontal positive of the grid elements is defined by the CADPTS.DAT file that lists the grid element number and x- and y-coordinates. With these two data files all the coordinate geometry (including elevation) of the entire grid system is defined. If the FPLAIN.DAT and CADPTS.DAT file were created with the GDS processor, these data files will be error free and no further modifications to these data files are necessary to start a simulation. The data files necessary to conduct a simple overland flow simulation are:

FPLAIN.DAT CADPTS.DAT CONT.DAT TOLER.DAT
INFLOW.DAT OUTFLOW.DAT

Overland flow velocities and depths vary with topography and the grid element roughness. Spatial variation in floodplain roughness can be assigned through the FLOENVIR processor. The assignment of overland flow roughness must account for vegetation, surface irregularity and flow path redirection. Overland roughness can be two or three times conventional open channel flow n-values and is a function of flow depth. Typical roughness values (Manning's n coefficients) for overland flow are shown in Table 2. An n-value of 0.070 is suggested for most overland flow applications.

Table 2. Overland Flow Manning's n Roughness Values[1]

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

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

IMPORTANT NOTE: Shallow overland flow (less 0.2 ft or 0.06 m) is defined by a shallow flow n-value (SHALLOWN) in the CONT.DAT file.

Overland flow depths and velocities will vary with grid element topography and roughness. The grid element attributes can be further modified to add detail to the predicted area of inundation. For example, the surface storage area or flow path on grid elements can be adjusted for buildings. Using the area reduction factors (ARFs), a grid element can be completely removed from receiving any inflow. Any of the eight flow directions can be partially or completely blocked to represent flow obstruction. The area of inundation can also be affected by levees, channel breakout flows, flow constriction at bridges and culverts, or street flow in urban areas. Rainfall and infiltration losses can add or subtract from the flow volume on the floodplain surface. These overland flow components are shown in a computational flow chart in Figure 7.

A number of FLO-2D project simulations have been created using the floodplain elements inside of the channel. In the case, the channel component is not used and instead the FLO-2D grid system is draped over the channel portion of the topography. While these projects have been conducted with some success, there are several modeling concerns that should be addressed. The FLO-2D model was developed to be able to exchange 1-D channel overbank discharge with the floodplain grid elements. For this reason, the model works well on large flood events and large grid elements. When small grid elements are used inside of a channel with confined flow and large discharges and flow depths, the model will run very slow. In addition, there will be variable bed topography that will result in zero water surface slope between some grid elements and spatially varied roughness should be used. It should be noted that the application of the open channel Manning's equation for a zero water surface slope is no long valid as the velocity approaches zero (ponded flow condition). Moving blocks of fluid volume with the channel grid elements in a finite timestep in 8 directions, has to result in variation of the water surface elevations in contiguous grid elements. The resulting water surface elevations can be accurately predicted but will display some variation of 0.1 ft (0.03 m) across the channel. Therefore, it is recommended, that the results for these in-channel detailed models be carefully reviewed. It is also important that the grid element size not exceed the resolution of the available DTM mapping data base.

IMPORTANT NOTE: A simple approach to grid element size selection is to use the criteria: $Q_{peak}/A_{surf} < 0.5$ cfs/ft^2 (0.15 m^3/m^2). This will result in reasonable timesteps on the order of about 1 second. If the value of $Q_{peak}/A_{surf} >> 1$ cfs/ft^2 (0.3 m^3/m^2), the model will run exceedingly slow. A_{surf} = surface area of a single grid element.

For overland flow, the specific energy, impact pressure and static pressure are computed and reported to file on an output interval basis (see the list of output files). The specific energy is computed by adding the flow depth velocity head ($V^2/2g$) to the flow depth. The maximum specific energy is reported to the file SPECENERGY.OUT by grid element. For some FEMA alluvial fan projects, the specific energy is to be reported and plotted as contours.

The impact pressure by floodplain grid element is reported as a force per unit length (impact pressure x flow depth). The user can then multiply the impact pressure by the structure length within the grid element to get a maximum impact force on the structure. Impact force is a function of fluid density, structure materials, angle of impact, and a number of other variables.

To conservatively estimate the impact pressure, the empirical equation for water is taken from Deng (1996):

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

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

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

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

The static pressure for each grid element is also expressed as a force per unit length. It is given by the maximum flow depth times the specific weight of the fluid. The static pressure is then multiplied by the flow depth to compute the static force per unit length of structure. The maximum static pressure is written to the STATICPRESS.OUT file.

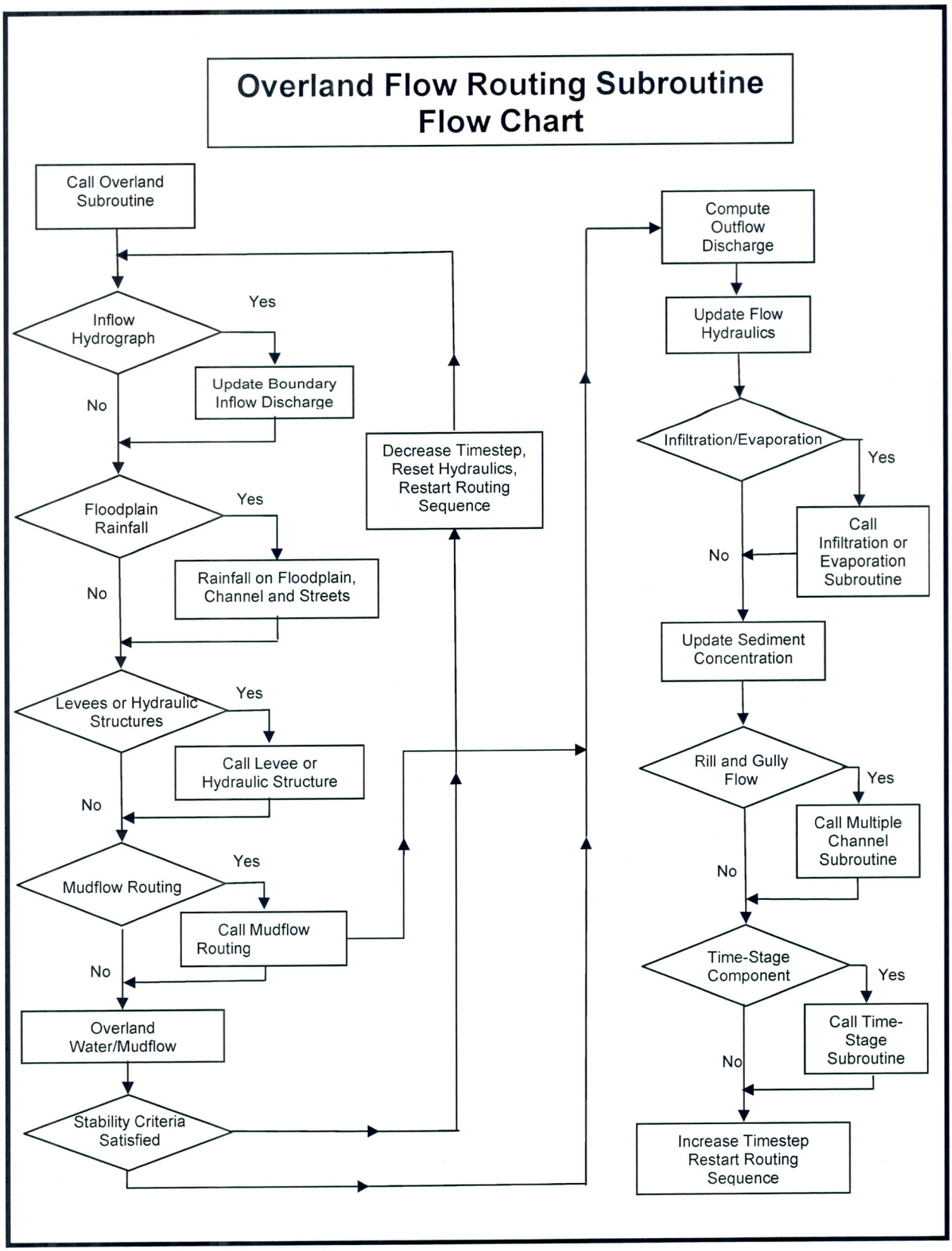


Figure 7. Overland Flow Routing Subroutine Flow Chart

3.3 Channel Flow

Channel flow is simulated as one-dimensional flow. Average flow hydraulics of velocity and depth define the discharge between channel grid elements. Secondary currents and dispersion are not modeled with a 1-D channel component. The flow around bridge piers or superelevation in channel bends cannot be simulated. The average flow path length between two channel elements is on the order of the length of the grid element and this precludes the simulation of hydraulic jumps over a short distance. The flow transition between subcritical and supercritical flow is based on the average conditions between two channel elements.

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

The channel width can be larger than the grid element and may encompass several elements (Figure 8). If the channel width is greater than the grid element width, the model extends the channel into neighboring grid elements in the direction specified by the user. A channel may be 1000 ft wide and the grid element only 300 feet square. The channel in this case would extend through three grid elements and the center grid in the channel would be conceptually removed from the floodplain. The model also makes sure that there is sufficient floodplain surface area after extension. The channel interacts with the right and left bank floodplain elements to share discharge. Each bank can have a unique elevation. If the two bank elevations are different in the CHAN.DAT file, the model automatically splits the channel into two elements even if the channel would fit into one grid element.

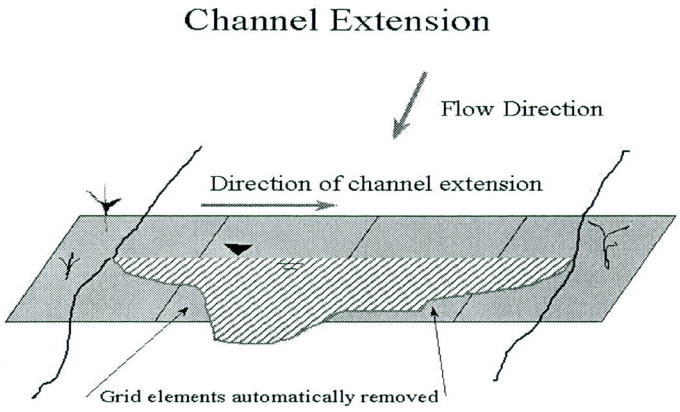


Figure 8. Channel Extension over Several Grid Elements

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

1. The channel grid element bed elevation is determined by subtracting the assigned channel thalweg depth from the floodplain elevation.

2. The channel bed elevation is assigned the grid element floodplain elevation and the thalweg depth is then added to the channel bed elevation to compute a new floodplain elevation.

3. A bank elevation is assigned in the CHAN.DAT file and the channel bed elevation is computed by subtracting the thalweg depth from the lowest bank elevation.

Generally, only options 1 and 3 are used. Option 2 reflects a system where a channel may be confined by berms and the floodplain is lower than the berm crest. When using cross section data for the channel geometry, option 3 should be applied.

The procedure for creating a river channel simulation is as follows:

1. *Select Channel Cross Sections.* Surveyed river cross sections can be represented in the model. These may be spaced to represent river reaches that encompass a number of grid elements, say 5 to 10 elements. Georeferenced surveyed cross section station and elevation data can be entered directly into the model data files or the data can be defined by setting the highest bank to an arbitrary elevation (e.g. 1000 ft). For channel design purposes, a rectangular or trapezoidal cross section may be selected. To use surveyed cross section data, a XSEC.DAT file has to be created with all cross section station and elevation data. Each channel element is then assigned a cross section in the CHAN.DAT. The relationship between the flow depth and channel geometry (flow area and wetted perimeter) is based on an interpolation of depth and flow area between vertical slices that constitute a rating table for each cross section.

   Another method to assign natural channel cross shape is to use the channel geometry routine with pre-assigned power regression relationships. The pre-processor program XSEC computes channel geometry relationships for the cross sections as a function of depth. This program XSEC transforms the cross section x- and y-coordinate data into the following relationships:

$$A = a\, d^b$$

$$P_w = a\, d^b$$

$$T_w = a\, d^b$$

Where A is the cross section flow area, P_w is the wetted perimeter, T_w is the top width, d is the depth, (a) is the regression coefficient and (b) is the regression exponent. More than one relationship may be used to define a single channel. For example, one relationship may be valid for flow depths less than 5 ft and a second relationship used to represent flows greater than 5 ft. This approach for representing the channel geometry in the FLO-2D is more computationally efficient than interpolating the flow from the cross section rating table data but is less accurate and is less numerically stable.

IMPORTANT NOTE: For river projects with available surveyed cross section data, the XSEC.DAT file is recommended over the channel geometry regressed relationships. The model will be less inclined to have channel surging at large timesteps.

2. *Locate the Channel Element with Respect to the Grid System.* By using the FLOENVIR processor program or by having a hard copy map with the grid system overlaid, the channels can be assigned to a grid element. For channel flow to occur through a reach of river, the channel elements must be neighbors. A channel extension direction can be defined and the channel length within the grid element can be assigned.

IMPORTANT NOTE: The channel length within a grid element can be estimated plus or minus 20 to 50 ft. When the CHAN.DAT is completed, the channel lengths can be summed in each reach using a spreadsheet program and compared to a centerline distance of the channel reach. The channel element lengths can then be adjusted so that the reach length is exact.

The FLO-2D model will automatically assign the grid elements necessary to contain the channel width. If the channel is to be contained within the grid element, the channel top width should not exceed 95% of the grid element width. There should be sufficient floodplain surface area (at least 5% of the total grid element surface area) for potential overbank flow storage. The model will extend the channel through enough grid elements to meet both of these conditions.

3. *Adjust the Channel Bed Slope and Interpolate the Cross Sections.* Each channel element is assigned a cross section in the CHAN.DAT file. Typically, there are only a few cross sections and many channel elements, so each cross section will be assigned to several channel elements. When the cross sections have all been assigned the channel profile looks like a stair case because the channel elements with the same cross section have identical bed elevations. The PROFILES processor programs allows you to view and edit the cross sections. There are two important functions that PROFILES will perform. First, by opening, saving and closing the PROFILES program, each channel element is assigned its own unique cross section. The number of cross sections is not equal to the number of channel elements and the cross section number in both XSEC.DAT and CHAN.DAT has been renumbered from top to bottom. Second, the cross sections and thalweg bed elevation can be interpolated. In PROFILES, the cross section editor has an interpolation button where the user can identify the upstream and downstream surveyed cross sections. Then the cross sections assigned to each channel element are interpolated between the two known cross sections. This accomplished as follows:

- Linear interpolation of the thalweg bed elevation.
- Linear interpolation of the channel top widths.
- Quasi-linear interpolation of stations and elevations for the revised top width.
- Weighted flow area adjustment of the cross section station and elevations to achieve a more uniform increase or decrease of flow area between the know cross sections.

The product of this effort is an assigned adjusted cross section and bed slope for each channel element. The assigned surveyed cross sections retain their original shape and elevations.

Rectangular and trapezoid cross sections can be edited directly in the CHAN.DAT file or using the FLOENVIR program. The bed slope for these two channel shapes can be adjusted in the PROFILES program. It is also possible to use the XSEC.DAT file to define rectangular or trapezoidal cross sections.

The user has several other options for setting up the channel data file including grouping the channel elements into segments, specifying initial flow depths, identifying contiguous channel elements that do not share discharge, assigning limiting Froude numbers and depth variable n-value adjustments. In river simulations the important components include channel routing, the channel-floodplain interaction, overland flow and levees. These components are described in more detail in the following sections

IMPORTANT NOTE: Manning's equation is an empirical formula that was developed on the basis of tests on steady, uniform, fully developed turbulent flow. Its use, however, has become universal for all flow applications. In the original derivation, the exponent of the hydraulic radius was determined to vary between 0.65 and 0.84. The use $R^{2/3}$ is only an approximation that must be balanced with varying n-values. There are occasions in a FLO-2D flood simulation when the flow is neither steady nor uniform. Floodplain ponded water or channel backwater effects are two instances when Manning's equation may not be appropriate. The tendency is to underestimate n-values in the FLO-2D model. Factors that affect roughness include surface roughness, obstructions, vegetation, sediment load, scour and deposition, channel size and shape, stage and discharge and rapidly varying flow (hydraulic jumps). Poor selection of n-values or failure to provide spatial variation in roughness can result in numerical surging. It is recommended to assign high n-values that represent a composite of the factors listed above. Avoid using n-values that represent prismatic channel flow.

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

3.4 Floodway Routine

A new floodway routine has been implemented into the FLO-2D system. The purpose of floodway concept is to reserve an unobstructed area of flood conveyance passage while allowing for potential utilization of the floodplain. Floodway boundaries are designed to accommodate a 100-yr flood within acceptable limits. The floodplain areas that can be eliminated from potential flood storage with violating the floodway criteria can be considered for potential development. The guidelines for floodway delineation are:

- The floodway is based on the 100-yr flood.
- The floodplain is divided into floodway and floodway fringe zones. It is generally assume that all the flood conveyance in the floodway fringe is eliminated.
- The floodway will pass the 100-yr flood without raising the water surface elevation more than 1 ft above the maximum floodplain water surface.
- The floodway is determined by means of equal reduction of conveyance on both sides of the channel.

The general procedure in HEC-RAS is to apply encroachment conditions using one or more of the encroachment options and make reasonable adjustments until acceptable results are obtained both from a flood hydraulics standpoint and from a floodplain management perspective. Floodway determination is difficult on streams with a mild slope and large floodplain, where there is split flow or overflow at drainage divides and levees, on alluvial channels with mobile boundaries, on high velocity channels and in developed floodplain areas with ineffective flow areas. One of major concerns is that the floodway encroachment procedure using HEC-RAS ignores the effects of both floodwave attenuation and the effects of forcing more flood volume downstream by constricting the flood conveyance area upstream. Using a single discharge model to delineate a floodway can grossly underestimate the potential impacts of increased downstream flooding resulting from permitting encroachment on the upstream floodplain.

The FLO-2D floodway component can address all the problems associated with floodplain encroachment. The procedure for identifying the floodway in FLO-2D is automated. First it is necessary to complete an existing conditions flood hazard delineation using the FLO-2D model on the project area. An output file (INTERGWS.OUT) is created that lists the maximum water surface elevations for each floodplain grid element. To perform the floodway analysis, the user sets the IFLOODWAY switch to "on" in CONT.DAT and assigns the encroachment depth (ENCROACH variable in CONT.DAT). Typically the encroachment depth is 1 ft. Then the FLO-2D model is re-run for the project. The model will add the encroachment depth to the maximum water surface elevation to compute an encroachment water surface elevation for a given grid element that must be exceeded in order for the model to exchange the discharge with other grid elements. As the overbank flooding ensues, the model confines the flood to those floodplain grid elements whose encroachment waste surface elevation is not exceeded. This forces more water volume downstream enhancing the opportunity to inundate the downstream floodplain in response to upstream confined conveyance. Both floodwave attenuation and upstream confined conveyances are simulated in a FLO-2D flood model. This provides a significant opportunity to evaluate the potential impacts of past delineated floodways.

3.5 Channel-Floodplain Interface

When the channel or street conveyance capacity is exceeded, an overbank discharge is computed in the channel-floodplain interface subroutine. The computed velocity of either the outflow from the channel or the return flow to the channel is based on the difference in water surface elevations between the channel and floodplain. If the channel flow is less than bankfull discharge and there is no flow on the floodplain, then the channel-floodplain interface routine is not accessed. The interface routine is internal to the model and there are no data requirements for its application. This subroutine also computes the flow exchange for the street and the floodplain.

The channel-floodplain exchange is based on the water surface elevation difference between the channel and the floodplain grid element containing either channel bank (Figure 2). The channel bank elevation is established by the surveyed channel geometry and thus the channel water surface and floodplain water surface is known in relationship to the channel top of bank. The water surface slope can vary from positive (flow out of the channel) to negative (return flow to the channel) on a timestep basis. Overbank discharge or return flow to the channel is computed using the diffusive wave equation and the floodplain roughness. The channel-floodplain flow exchange is limited by the available exchange volume in the channel or by the available storage volume on the floodplain. This interface determines how much overbank discharge will be routed as unconfined floodplain flow. An inflow hydrograph node can be located on the floodplain and the overland flow can enter a previously dry channel.

3.6 Limiting Froude Numbers

The Froude number has a number of physical implications; it delineates subcritical and supercritical flow, it is the ratio of average flow velocity to shallow wave celerity and it relates the movement of a translational wave to stream flow. Establishing a limiting Froude number in a flood routing model can help sustain the numerical stability by forcing the model to have a reasonable representation this physical reality. In alluvial river channel flow, the practical range of Froude numbers at bankfull discharge is 0.4 to 0.6. Overland flow on steep alluvial fans can approach critical flow. In general, supercritical flow on alluvial fans is suppressed by high rates on sediment transport. High velocities and shallow depths on alluvial surfaces will dissipate energy with sediment entrainment. Supercritical flow is more prevalent on bedrock or other hard surfaces.

When a limiting Froude is assigned for either floodplain flow, street flow or channel flow for a given reach, the model computes the grid element flow direction Froude number for each timestep. If the limiting Froude number is exceeded, the Manning's n-value for hydraulic flow resistance is increased by 0.001. As the flow slows down, the n-value is decreased by 0.0001. This increase in flow resistance mimics increasing energy loss as the flow accelerates. When the limiting Froude is exceeded, the changes in the n-value are reported in the ROUGH.OUT file.

There is a unique relationship that exists between slope, flow area and roughness. If there is a mismatch between these physical variables in a flood routing model, then high velocities can occur that may result in flow surging. Assigning a limiting Froude number has several practical advantages. First, it helps to maintain the flow velocity within a reasonable range. Secondly, a review of the increased n-values in ROUGH.OUT will identify any trouble spots where the velocity exceeds a reasonable value. In this case, the roughness value is increased to offset an inappropriate flow area and slope relationship. The n-values in CHAN.DAT and FPLAIN.DAT can be adjusted for the next simulation using the maximum values report in the ROUGH.OUT file. Finally, the increased n-values can prevent oversteepening of the frontal wave. The limiting Froude number is assigned in CONT.DAT for the floodplain, CHAN.DAT for channel flow and STREET.DAT for street flow.

IMPORTANT NOTE: It is suggested that the channel bankfull flow Froude number be estimated for a given channel reach or segment. This would constitute the limiting channel Froude number assigned in the CHAN.DAT file. For an alluvial fan analysis, a limiting Froude number of 0.90 to 0.98 is recommended. For street flow, the limiting Froude number may be supercritical and should be estimated. A practical range of the limiting Froude number is 1.2 to 1.5.

3.7 Levees

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

Levee failure is an option. Simulated levee failure can occur with a breach that enlarges vertically or horizontally. Rates of breach expansion in feet or meters per hour can be specified for both the horizontal and vertical failure modes. A final levee base elevation that is higher than the floodplain elevation can also be specified. The entire grid element width for a given flow direction can fail. Discharge through the breach is based on the levee width and the difference in water surface elevations on the two sides of the levee. Levee failure can also be initiated by flood duration at a specified water surface elevation on the levee.

IMPORTANT NOTE: The levee can be easily created and edited graphically in the GDS and FLOENVIR processor programs. When the levee is situated diagonally across a series of grid elements, use one side of one grid element and the opposite of the next grid element so that there an equal amount of the floodplain on each side of the levee.

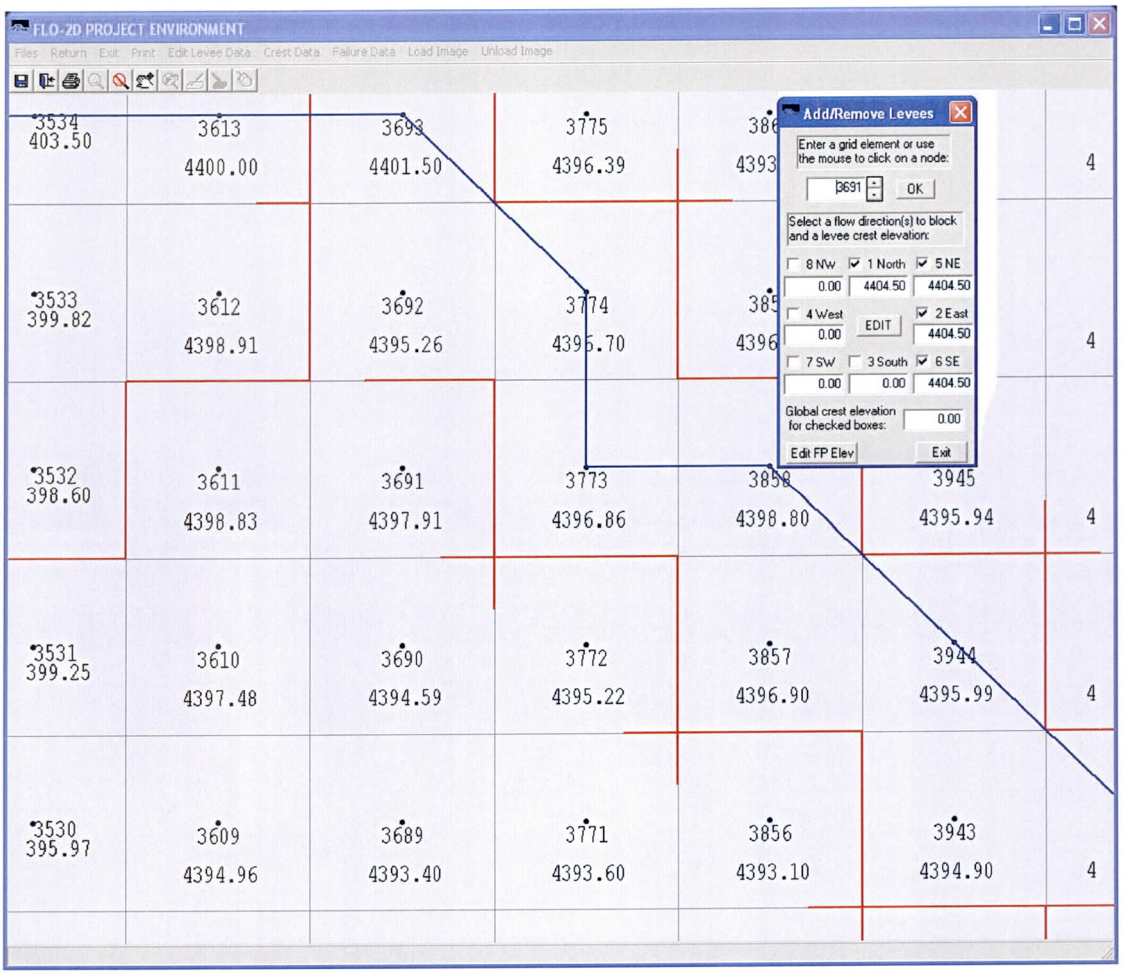


Figure 9. Levees Depicted in Red and River in Blue in the FLOENVIR Program

3.8 Hydraulic Structures

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

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

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

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

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

3.9 Street Flow

Street flow is simulated as flow in shallow rectangular channels with a curb height using the same routing algorithm as channels. The data input file STREET.DAT is organized by street. A given grid element may contain one or more streets and the streets may intersect. The user specifies a street name followed by the number of grid elements that constitute a given section of street. The flow direction, street width and roughness are specified for each street section within the grid element. Street and overland flow exchanges are computed in the channel-floodplain flow exchange subroutine. When the curb height is exceeded, the discharge to floodplain portion of the grid element is computed. Return flow to the streets is also simulated.

Streets are assumed to emanate from the center of the grid element to the element boundary in the eight flow directions (Figure 10). For example, an east-west street across a grid element would be assigned two street sections. Each section has a length of one-half the grid element side or diagonal. A given grid element may contain one or more streets and the streets may intersect. Street roughness values, street widths, elevations and curb heights can be modified on a grid element or street section basis in the FLOENVIR program.

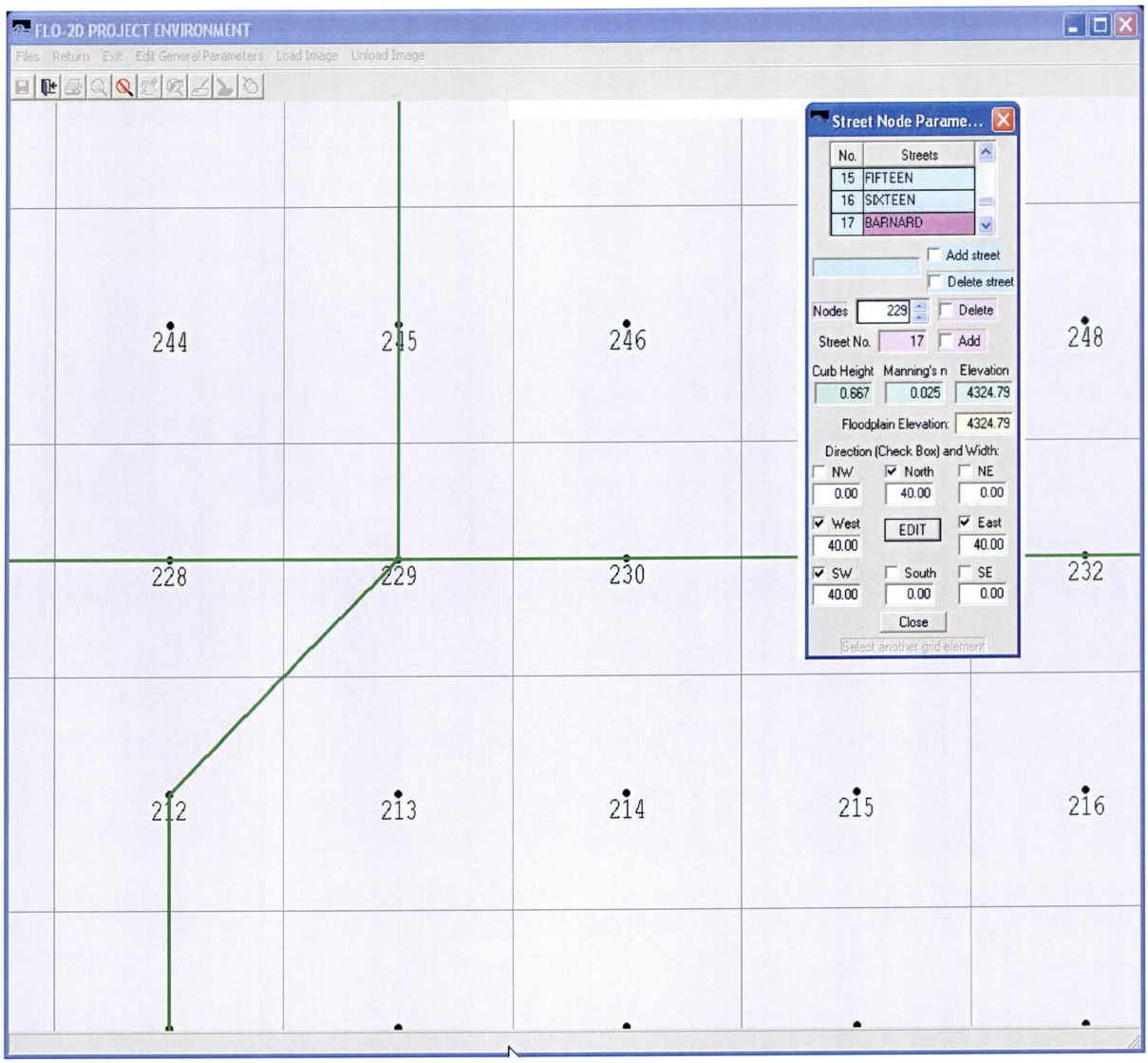


Figure 10. Streets Depicted in Green in the FLOENVIR Program.

3.10 Floodplain Surface Storage Area Modification

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

These factors can greatly enhance the detail of the flood simulation through an urban area. Area reduction factors are specified as a percentage of the total grid element surface area (less than or equal to 1.0). Width reduction factors are specified as a percentage of the grid element side (less than or equal to 1.0). For example, a wall might obstruct 40% of the flow width of a grid element side and a building could cover 75% of the same grid element (Figure 11).

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

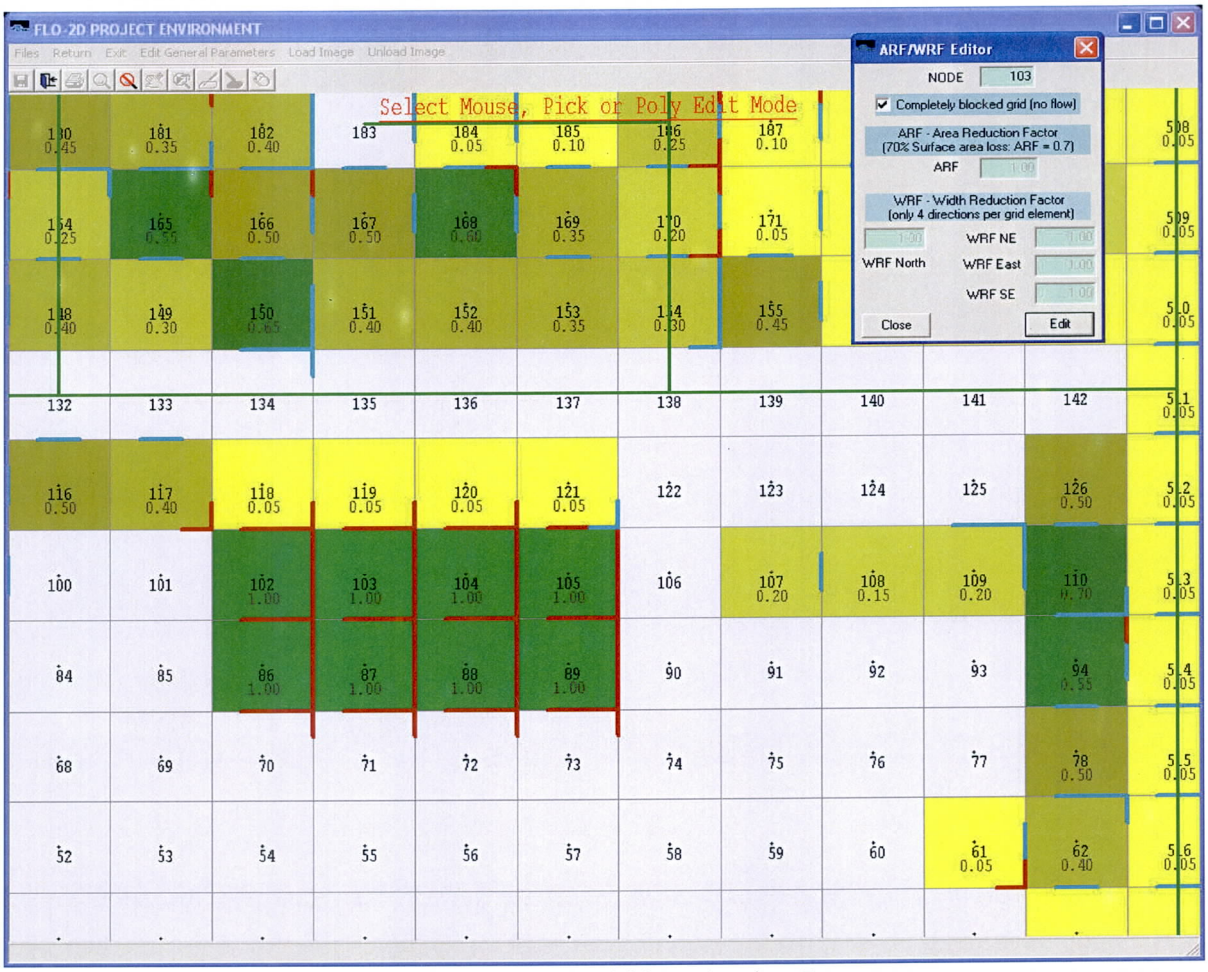


Figure 11. Area and Width Reduction Factors

3.11 Rainfall

Alluvial fan or floodplain rainfall can make a substantial contribution to the flood volume and peak discharge. Some the fan or floodplain surface areas are of the same order of magnitude as the upstream watershed area. In these cases excess rainfall on the fan or floodplain may equal or exceed the total volume of inflow hydrograph from the watershed. The excess rainfall and runoff on the fan or floodplain can precede the arrival of the floodwave from the upstream watershed. It will also dilute the mudflows from the upstream basin.

The storm rainfall is discretized as a cumulative percent of the total precipitation (similar to HEC-1 data files) for input to the model in the RAIN.DAT file. This discretization of the storm hyetograph is established through local rainfall data or through regional drainage criteria that defines storm duration, intensity and distribution. Storm characteristics can significantly affect the excess runoff when simulating infiltration and a careful review of local published rainfall/runoff data is encouraged to establish the rainfall distribution. The first rainfall timestep may correspond to the first upstream flood inflow hydrograph timestep. By altering the storm time distribution on the fan or floodplain, the rainfall can lag or precede the rainfall in the upstream basin depending on the direction of the storm movement over the basin. The storm can also have more or less total rainfall than that occurring in the upstream basin. If no infiltration is being modeled, the user can still specify an initial abstraction.

Storms can be varied spatially over the grid system with areas of intense or light rainfall. The rainfall can be assigned on a grid element basis using real rainfall data. FLOENVIR can used to draw the storm shape and assign rainfall intensity factors. Storms can also move over the grid system by assigning storm speed and direction.

3.12 Infiltration and Abstraction

Precipitation losses simulated in the FLO-2D model include abstraction (interception) and infiltration. The initial abstraction is filled prior to simulating infiltration and is assigned by the user in the INFIL.DAT file. Some typical initial abstraction values are presented in Table 3.

Infiltration is simulated using the Green-Ampt infiltration model. Spatial variation of infiltration over the grid system can be modeled by assigning unique hydraulic conductivity and soil suction values to each grid element. The infiltration parameters are assumed to be uniformly distributed over a grid element surface with the exception that no infiltration is calculated for streets, buildings or other impervious surfaces. The infiltration parameters are usually assigned a uniform value over a subbasin unless there is field data to support a detailed spatial variation. Spatially variable infiltration can be graphically assigned in the FLOENVIR program. Channel infiltration can be simulated by setting the INFIL switch in the INFIL.DAT file. Although channel infiltration and bank seepage are generally very minor portion of the total infiltration losses in the system, they can affect the floodwave progression in an ephemeral channel. The surface area of a natural channel is used to approximate the wetted perimeter to compute the infiltration volume.

Table 3. Initial Abstraction	
Surface Cover	Abstraction (inches)
Natural[1]	
Desert and rangeland	0.35
Hillslopes Sonoran desert	0.15
Mountain with vegetation	0.25
Developed – Residential[1]	
Lawns	0.20
Desert landscape	0.10
Pavement	0.05
Agricultural fields and pasture	0.50
Conifers[2]	0.01 - 0.36
Hardwoods[2]	0.001 - 0.08
Shrubs[2]	0.01 - 0.08
Grass[2]	0.04 - 0.06
Forest floor[2]	0.02 - 0.44
[1]Maricopa County Drainage Design Manual, 1992. [2]W. T. Fullerton, Masters Thesis, CSU, 1983	

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

- Air displacement from the soil has a negligible effect on the infiltration process.
- Infiltration is a vertical process represented by a distinct piston wetting front.
- Soil compaction due to raindrop impact is insignificant.
- Hysteresis effects of the saturation and desaturation process are neglected.
- Flow depth has limited effect on the infiltration processes.

A derivation of the Green-Ampt infiltration modeling procedure can be found in Fullerton (1983) and in the HEC-1 Manual (USCOE, 1990). To utilize the Green-Ampt model, hydraulic conductivity, soil suction and the volumetric moisture deficiency must be specified in the INFIL.DAT file. Typical hydraulic conductivity, porosity and soil suction parameters are presented in Tables 4 and 5. The volumetric moisture deficiency is evaluated as the difference between the initial and final soil saturation conditions. The initial saturation can be determined by subtracting the volumetric moisture deficiency (Table 6) from the final saturation (often assumed to be 1). Depression storage is an initial loss from the potential surface flow. This is the amount of water in small surface depressions that does not become part of the overland runoff or infiltration. The depression storage or head (ft) is assigned by the TOL variable in the TOLER.DAT. A typical value of TOL is 0.1 ft. No discharge flux is computed for flow depths less than TOL.

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

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

Table 5. Green Ampt Infiltration Parameters - Soil Suction			
Classification	(in)[1]	(in)[2]	(in)[3]
sand and loamy sand	2.4	1.9-2.4	
sandy loam	4.3	4.3	
loam	3.5	3.5	
silty loam	6.6	6.6	
silt	7.5		
sandy clay loam	8.6	8.6	
clay loam	8.2	8.2	
silty clay loam	10.8	10.8	
sandy clay	9.4	9.4	
silty clay	11.5	11.5	
clay	12.4	12.5	
Nickel gravel-sand loam			2.0 - 4.5
Ida silt loam			2.0 - 3.5
Poudre fine sand			2.0 - 4.5
Plainfield sand			3.5 - 5.0
Yolo light clay			5.5 - 10.0
Columbia sandy loam			8.0 - 9.5
Guelph loam			8.0 - 13.0
Muren fine clay			15.0 - 20.0

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

Table 6. Green Ampt Infiltration Parameters – Volumetric Moisture Deficiency		
Classification	Dry (% Diff)	Normal (% Diff)
sand and loamy sand[1]	35	30
sandy loam	35	25
loam	35	25
silty loam	40	25
silt	35	15
sandy clay loam	25	15
clay loam	25	15
silty clay loam	30	15
sandy clay	20	10
silty clay	20	10
clay	15	5
[1]Maricopa County Drainage Design Manual, 1992.		

3.13 Evaporation

An open water surface evaporation routine was created for the FLO-2D model to account for evaporation losses in major river systems for long duration flood flows. This component was implemented for the 173 mile Middle Rio Grande model from Cochiti Dam to Elephant Butte Reservoir. The open water surface evaporation computation is based on a total monthly evaporation that is prorated for the number of flood days in the given month. The user must input the total monthly evaporation in inches or mm for each month along with the presumed diurnal hourly percentage of the daily evaporation and the clock time at the start of the flood simulation. The total evaporation is then computed by summing the wetted surface area on both the floodplain and channel grid elements for each timestep. The floodplain wetted surface area excludes the area defined by ARF area reduction factors. The evaporation loss does not include evapotranspiration from floodplain vegetation. Evapotranspiration is a function of the groundwater/surface water interaction and should be considered in relationship to the seepage (infiltration) loss and change in groundwater storage. The total evaporation loss is reported in the SUMMARY.OUT file and should be compared with the infiltration loss for reasonableness.

3.14 Overland Multiple Channel Flow

The purpose of the multiple channel flow routine is to simulate the overland flow in rills and gullies rather than as overland sheet flow. Watershed surface water is often conveyed in small gully channels, even though they occupy only a fraction of the potential flow area. Simulating rill and gully flow concentrates the discharge and improves the timing of overland runoff routing. The multiple channel routine calculates overland flow as sheet flow and routes it to the multiple channels within the grid element. Flow between the grid elements is then computed as rill and gully flow. No overland sheet flow is exchanged between grid elements if both elements have assigned multiple channels in the MULT.DAT file. Discharge can be conveyed as rill and gully flow from a grid with multiple channels to a grid element without them. The gully geometry is defined by a maximum depth, width and flow roughness. The

multiple channel attributes can be spatially variable on the grid system and can be edited with the FLOENVIR program.

If the gully flow exceeds the specified gully depth, the multiple channel can be expanded by a specified incremental width. This channel widening process assumes these gullies are alluvial channels and will widen to accept more flow as the flow reaches bankfull discharge. There is no gully overbank discharge to the overland surface area within the grid element. The gully will continue to widen until the gully width exceeds the width of the grid element, then the flow routing between grid elements will revert to sheet flow. This enables the grid element to be overwhelmed by flood flows. During the falling limb of the hydrograph when the flow depth is less than 1 ft., the gully width will decrease to confine the discharge until the original width is again attained. If a grid element contains streets or river channels in addition to multiple channels such that the available surface area is less than 20% of the original grid element surface area, then the program will eliminate the gullies and reset that particular element to overland sheet flow.

3.15 Sediment Transport – Total Load

FEMA FIS studies are usually conducted using a rigid bed hydraulic model such as the Corps of Engineers HEC-RAS model. The National Research Council (1983) evaluated several movable-bed models without recommending any specific model for predicating channel geometry changes. When a channel rigid bed analysis is performed, any potential cross section changes are assumed to have a negligible effect on the predicted water surface. This is a reasonable assumption for large flood events on the order of a 100-year flood. On steep alluvial fans, several feet of scour or deposition will usually have a minimal effect on the flow paths of large flood events. For small flood events, the potential effects of channel incision, avulsion, blockage, bank or levee failure and sediment deposition on the flow path should be considered.

To address mobile bed issues, FLO-2D has a sediment transport component that can compute sediment scour or deposition. Within a given grid element, sediment transport capacity is computed for either channel flow or overland flow based on the flow hydraulics. The sediment transport capacity is then compared with the sediment supply and the resulting sediment excess or deficit is uniformly distributed over the grid element potential flow surface. The sediment transport capacity is computed using a choice of six possible equations for alluvial channels including Zeller and Fullerton (1983), Yang's equation, Ackers and White, Englund and Hansen, Laursen or Toffeletti. Each sediment transport formula was derived for unique fluvial geomorphic conditions and the user is encouraged to research the applicability of a selected equation to each project.

FLO-2D can compute total load sediment transport in channels, streets and overland flow. The user has a choice of six sediment transport capacity methods and equations including Zeller-Fullerton, Yang's, Ackers and White, Engelund and Hansen, Laursen and Toffaleti. Each sediment transport equation was developed to simulate specific channel or bed material conditions and has unique attributes that may limit their applicability to certain river reaches.

The user is encouraged to investigate each sediment transport equation and determine whether a given equation is appropriate for their modeling project.

In the model, the sediment transport equation is used to compute the sediment transport capacity based on the predicted flow hydraulics between grid or channel elements. Sediment continuity is tracked through the system on a grid element basis. The computed sediment load out of a grid element is compared to the sediment supply from other grid elements and the difference is deposited or scour from the grid element bed. The sediment excess (deposition) or deficit (erosion) is distributed uniformly on the floodplain element or non-uniformly on channel cross section based on a bed porosity of 0.40. The maximum scour, deposition and final bed elevations are recorded in the SEDFP.OUT and SEDCHAN.OUT files.

Sediment transport is uncoupled from flow hydraulics. First, the flow hydraulics are computed for all the grid and channel elements for the given time step and then the sediment transport is computed based on the completed flow hydraulics for that timestep. This assumes that the change in channel geometry resulting from deposition or scour does not have a significant effect on the average flow hydraulics for that timestep. Generally it takes several timesteps on the order of 10 seconds to result in average sediment deposition or scour that exceeds 0.05 ft.

Each sediment transport equation is briefly described. The user is encouraged to do further research to determine which equation is most appropriate for a specific project bed material and channel hydraulics. It should be noted that each equation may have been significant limitations that should be observed. When reviewing the SEDTRANS.OUT file, it can be observed that the Ackers-White and Engelund-Hansen equations generate the highest sediment transport capacity; Yang and Zeller-Fullerton result in a moderate sediment transport quantities; and Laursen and Toffaleti compute the lowest sediment transport capacity. The apply any of the equations, the ISED switch in the CONT.DAT file must be turned "on" and for channel flow the ISEDN switch must be set equal to 1 in CHAN.DAT file for each channel segment.

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

Engelund-Hansen Method. Bagnold's stream power concept was applied with the similarity principle to derive a sediment transport function. The method involves the energy slope, velocity, bed shear stress, median particle diameter, specific weight of sediment and water, and gravitational acceleration. In accordance with the similarity principle, the method

should be applied only to flow over dune bed forms, but Engelund and Hansen (1967) determined that it could be effectively used in both dune bed forms and upper regime sediment transport (plane bed) for particle sizes greater than 0.15 mm.

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

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

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

Zeller-Fullerton Equation. Zeller-Fullerton is a multiple regression sediment transport equation for sand bed channels or alluvial floodplains is used in the model. This empirical equation is a computer generated solution of the Meyer-Peter, Muller bed-load equation applied in conjunction with Einstein's suspended load integration (Zeller and Fullerton, 1983). The bed material discharge q_s is calculated in cfs per unit width as follows:

$$q_s = 0.0064\ n^{1.77}\ V^{4.32}\ G^{0.45}\ d^{-0.30}\ D_{50}^{-0.61}$$

where n is Manning's roughness coefficient, V is the mean velocity, G is the gradation coefficient, d is the hydraulic depth and D_{50} is the median sediment diameter. All units in this

equation are in the ft-lb-sec system except D_{50}, which is in millimeters. If the metric option is activated, no unit conversions are necessary.

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

Summary. Yang (1996) makes the following recommendations for the application of total load sediment transport formulas in the absence of measured data. The recommendations have been limited to the six equations provided in the FLO-2D model and are slightly edited:

- Use Meyer-Peter and Muller and Einstein procedure (Zeller and Fullerton equation) when the bedload is a significant portion of the total load.
- Use Toffaleti's method for large sand-bed rivers.
- Use Yang's equation for sand and gravel transport in natural rivers.
- Use Ackers-White or Engelund-Hansen equations for subcritical flow in lower sediment transport regime.
- Use Lausen's formula for shallow rivers with silt and fine sand.

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

It is important to note that in applying these equations, the wash load is not included in the computations. Therefore, the wash load should be subtracted from any field measurements before comparing with the predicted sediment transport results from the equations. It is also important to recognize if the field measurements are supply limited. In this case, comparison with the sediment transport capacity equations would be inappropriate.

3.16 Mud and Debris Flow Simulation

FLO-2D routes hyperconcentrated sediment flows (mud and debris flows) as a fluid continuum by predicting viscous fluid motion. For mudflows, the motion of the fluid matrix is governed by the sediment concentration. A quadratic rheologic model for predicting viscous and yield stresses as function of sediment concentration is employed and sediment volumes are tracked through the system. As sediment concentration changes for a given grid element, dilution effects, mudflow cessation and the remobilization of deposits are simulated. Mudflows are dominated by viscous and dispersive stresses and constitute a very different phenomenon than those processes of suspended sediment load and bedload in conventional sediment transport. The sediment transport and mudflow components **cannot** be used together in a FLO-2D simulation.

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

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

O'Brien and Julien (1988), Major and Pierson (1990), and Julien and Lan (1991) investigated mudflows with high concentrations of fine sediment in the fluid matrix. These studies showed that mudflows behave as Bingham fluids with low shear rates ($<10\ s^{-1}$). In fluid matricies with low sediment concentrations, turbulent stresses dominate in the core flow. High concentrations of coarse particles combined with low concentrations of fine particles are required to generate dispersive stresses. The quadratic shear stress model proposed by O'Brien and Julien (1985) describes the continuum of flow regimes from viscous to turbulent/dispersive flow.

To route mudflows, the rheological behavior of the flow must be treated as a continuum with mixed water and sediment components. Hyperconcentrated sediment flows such as mud and debris flows involve the interaction of complex fluid and sediment processes; turbulence, viscous shear, fluid-sediment particle momentum exchange, particle drag and sediment particle collision. Sediment particles can collide, grind, and rotate in their movement past each other. Fine sediment cohesion controls the nonNewtonian behavior of the fluid matrix. This cohesion contributes to the yield stress τ_y which must be exceeded by an applied stress in order to initiate fluid motion. By combining the yield stress and viscous stress components, the well-known Bingham plastic model is prescribed.

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

dramatically and as a result viscous and turbulent stresses may alternately dominate, producing flow surges.

The shear stress in hyperconcentrated sediment flows, including those described as debris flows, mudflows and mud floods, can be calculated from the summation of five shear stress components.

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

where the total shear stress τ depends on the cohesive yield stress τ_c, the Mohr-Coulomb shear τ_{mc}, the viscous shear stress τ_v (η dv/dy), the turbulent shear stress τ_t, and the dispersive shear stress τ_d. When written in terms of shear rates (dv/dy) the following quadratic rheological model can be developed (O'Brien and Julien, 1985):

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

where

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

and

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

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

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

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

$$f(\rho_s, \, C_v \,) = \frac{\pi}{12} \left(\frac{6}{\pi} \right)^{1/3} \sin^2 \alpha_1 \;\; \rho_s (1 - e_n^2 \,) \, C_v^{1/3}$$

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

The first two stress terms in the shear stress equation are referred to as the Bingham shear stresses and represent the internal resistance stresses of a Bingham fluid (Figure 6). The sum of the yield stress and viscous stress defines the shear stress of a cohesive, hyperconcentrated sediment fluid in a viscous flow regime. The last term is the sum of the dispersive and turbulent shear stresses, which is a function of the square of the velocity gradient. A discussion of these stresses and their role in hyperconcentrated sediment flows can be found in Julien and O'Brien (1987, 1993).

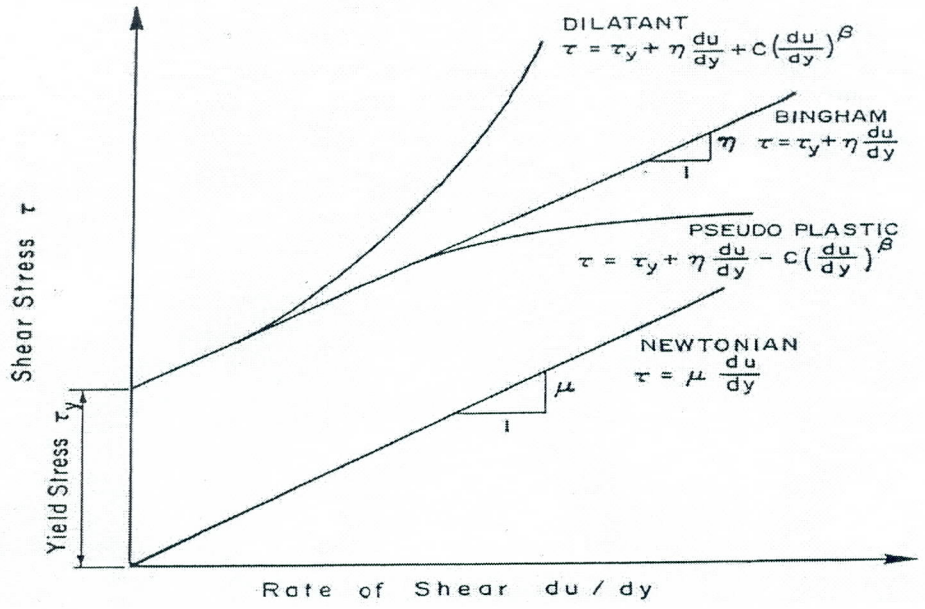


Figure 12. Shear Stress as a Function of Shear Rate for Fluid Deformation Models

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

To delineate the role turbulent and dispersive forces in sand and water mixtures, Hashimoto (1997) developed a simplified criteria involving only flow depth (d) and sediment size D_i. When $d/D_i < 30$, the intergranular forces are dominant. If $d/D_i > 100$, inertial forces dominate. In the range $30 < d/D_i < 100$ both forces play an important role in momentum

exchange. It should be noted, however, that sediment concentration is a critical factor that is not accounted for in this criteria.

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

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

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

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

where γ_m is the specific weight of the sediment mixture. The resistance parameter K for laminar flow equals 24 for smooth wide rectangular channels but significantly increases (up to 50,000) with roughness (vegetation) and irregular cross section geometry (Table 7). For Kentucky Blue Grass with a slope of 0.01, K was estimated at 10,000 (Chen, 1976). A value of K = 2,285 was calibrated on the Rudd Creek, Utah mudflow for a residential area and has been used effectively in most studies. For laminar and transitional flows, turbulence is suppressed and the laminar flow resistance parameter K becomes important.

Table 7. Resistance Parameters for Laminar Flow[1]

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

[1] Woolhiser (1975)

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

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

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

$$n_d = n_t\, b\, e^{mC_v}$$

where n_t is the turbulent n-value, b is a coefficient (0.0538) and m is an exponent (6.0896). This equation was based on …

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

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

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

$$\eta = \alpha_1\ e^{\beta_1 C_v}$$

and

$$\tau_y = \alpha_2\ e^{\beta_2 C_v}$$

where α_i and β_i are empirical coefficients defined by laboratory experiment (O'Brien and Julien, 1988). The viscosity and yield stress are shown to be functions of the volumetric sediment concentration C_v of silts, clays and in some cases, fine sands and do not include larger clastic material rafted along with the flow (Table 8 and Figs. 7 and 8).

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

Source	$\eta = \alpha e^{\beta Cv}$		$\tau_y = \alpha e^{\beta Cv}$	
	α	β	α	β
Field Data				
Aspen Pit 1	0.181	25.7	0.0360	22.1
Aspen Pit 2	2.72	10.4	0.0538	14.5
Aspen Natural Soil	0.152	18.7	0.00136	28.4

Aspen Mine Fill	0.0473	21.1	0.128	12.0
Aspen Watershed	0.0383	19.6	0.000495	27.1
Aspen Mine Source Area	0.291	14.3	0.000201	33.1
Glenwood 1	0.0345	20.1	0.00283	23.0
Glenwood 2	0.0765	16.9	0.0648	6.20
Glenwood 3	0.000707	29.8	0.00632	19.9
Glenwood 4	0.00172	29.5	0.000602	33.1
Relationships Available from the Literature				
Iida (1938)	-	-	0.0000373	36.6
Dai et al. (1980)	2.60	17.48	0.00750	14.39
Kang and Zhang (1980)	1.75	7.82	0.0405	8.29
Qian et al. (1980)	0.00136	21.2	-	-
	0.050	15.48	-	-
Chien and Ma (1958)	0.0588	19.1-32.7	-	-
Fei (1981)	0..166	25.6	-	-
	0.00470	22.2	-	-

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

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

The viscosity of hyperconcentrated sediment flow is a function of the properties of the fluid matrix including sediment concentration, percent and type of silts and clays and fluid temperature. Viscous mudflows have high sediment concentrations and correspondingly high yield stresses and may result in laminar flow. Less viscous flows (mud floods) are always turbulent. The mudflow sample parameters in Table 8 represent a full range of potential flow characteristics.

To simulate mudflows with the FLO-2D model, the MUD switch in the CONT.DAT must be set and the viscosity and yield stress variables in SED.DAT file must be specified. It is recommended that the viscosity and yield stress exponents and coefficients from Table 8 be selected for inclusion in the SED.DAT file. The sample Glenwood 4, for example, creates a very viscous mudflow. A volumetric sediment concentration or a sediment volume must then be assigned to the water discharge for a timestep in the discretized inflow hydrograph in the INFLOW.DAT file. The inflow sediment volume may represent channel scour, bank erosion or hillslope failure. The incremental sediment volume is tracked through the routing simulation and reported as a total sediment volume in the summary volume conservation tables. This total sediment volume should be reviewed to determine if this is a reasonable sediment yield for the watershed.

For a mudflow event, the average sediment concentration generally ranges between 20% and 35% by volume with peak concentrations approaching 45% (Table 9 and Figure 9). Large

flood events such as the 100-year flood may contain too much water to produce viscous mudflow event. Smaller rainfall events such as the 10- or 25-year return period storm may have a greater propensity to create viscous mudflows. Most watersheds with a history of mud and debris flow events will have a substantial sediment supply. Even very small storms may generate mudflow surges. Mudflow and debris flows on semi-arid southwest alluvial fans usually average less than 15% concentration by volume. The areal extent of mudflow inundation and the maximum flow depths and velocities can be determined by varying the sediment concentrations in the inflow hydrograph. A few trials will quickly illustrate the effect of changing the sediment concentration.

Most mudflows have a distinct pattern of flood development. Initially, clear water flows from the basin rainfall-runoff may arrive at the fan apex. This may be followed by a surge or frontal wave of mud and debris (40 to 50% concentration by volume). When the peak arrives, the average sediment concentration generally decreases to the range of 30 to 40% by volume. On the falling limb of the hydrograph, surges of high concentration may occur.

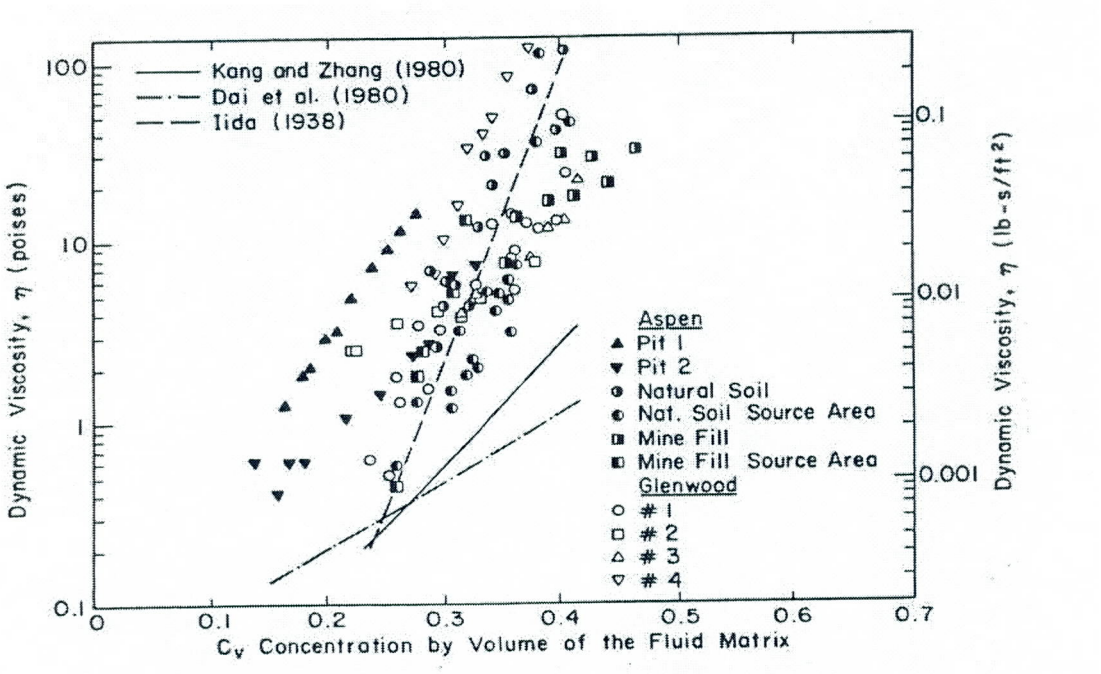


Figure 13. Dynamic Viscosity of Mudflow Samples Versus Volumetric Concentration

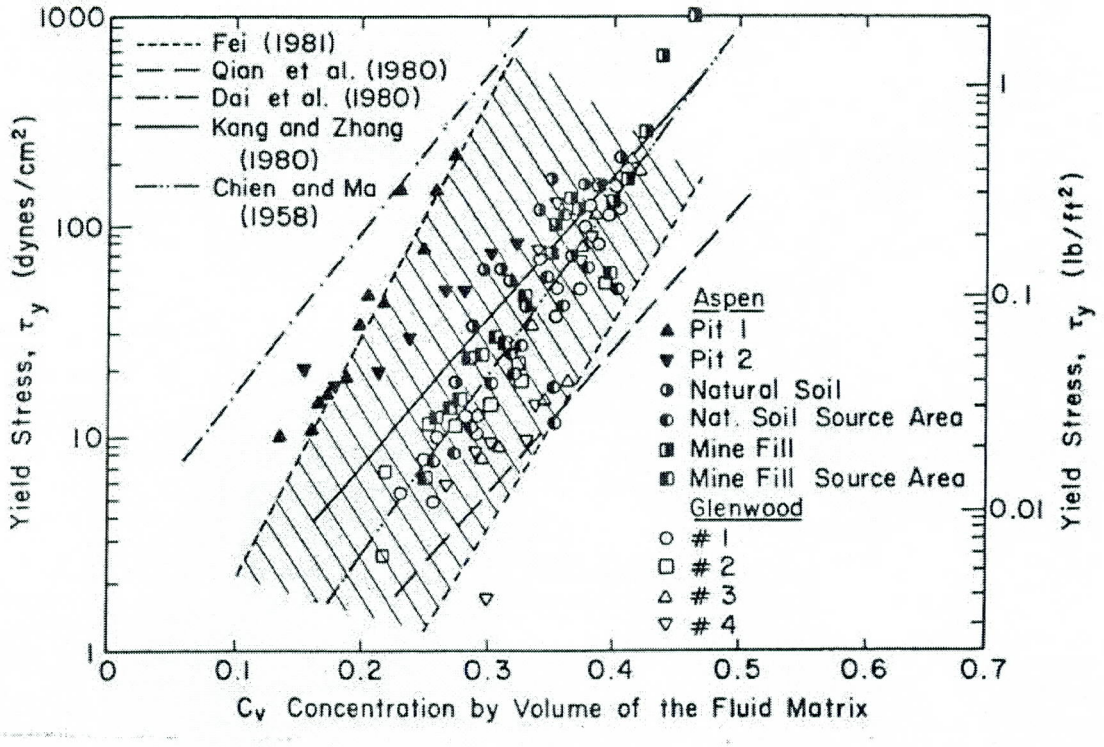


Figure 14. Yield Stress of Mudflow Samples Versus Volumetric Concentration

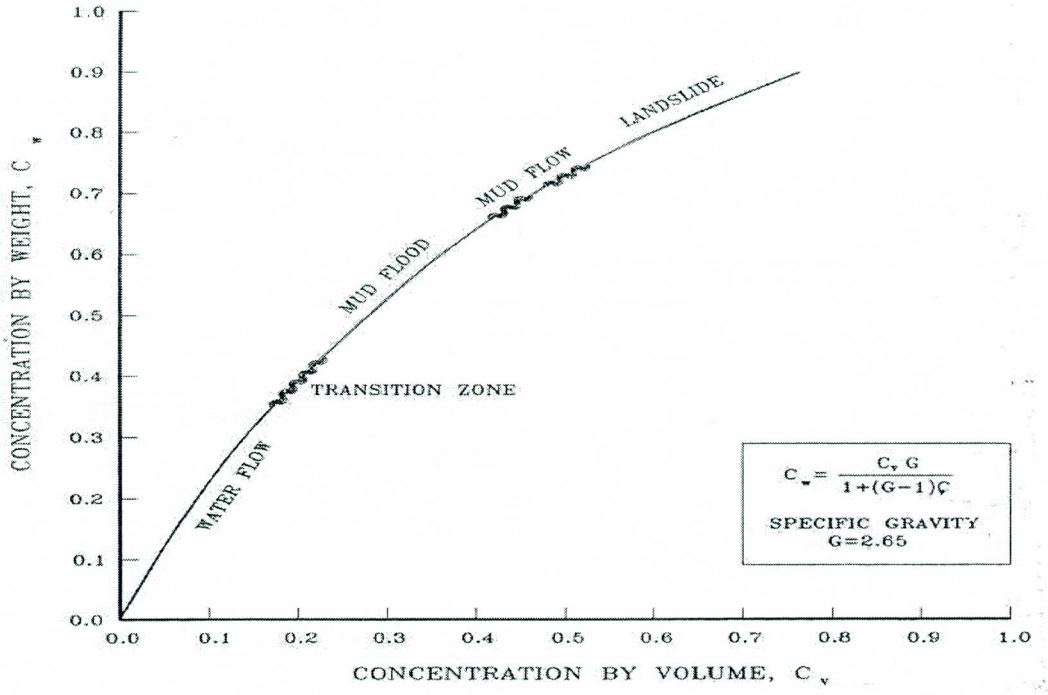


Figure 15. Classification of Hyperconcentrated Sediment Flows

Table 9. Mudflow Behavior as a Function of Sediment Concentration			
	Sediment Concentration		Flow Characteristics
	by Volume	by Weight	
Landslide	0.65 - 0.80	0.83 - 0.91	Will not flow; failure by block sliding
	0.55 - 0.65	0.76 - 0.83	Block sliding failure with internal deformation during the slide; slow creep prior to failure
Mudflow	0.48 - 0.55	0.72 - 0.76	Flow evident; slow creep sustained mudflow; plastic deformation under its own weight; cohesive; will not spread on level surface
	0.45 - 0.48	0.69 - 0.72	Flow spreading on level surface; cohesive flow; some mixing
Mud Flood	0.40 - 0.45	0.65 - 0.69	Flow mixes easily; shows fluid properties in deformation; spreads on horizontal surface but maintains an inclined fluid surface; large particle (boulder) setting; waves appear but dissipate rapidly
	0.35 - 0.40	0.59 - 0.65	Marked settling of gravels and cobbles; spreading nearly complete on horizontal surface; liquid surface with two fluid phases appears; waves travel on surface
	0.30 - 0.35	0.54 - 0.59	Separation of water on surface; waves travel easily; most sand and gravel has settled out and moves as bedload
	0.20 - 0.30	0.41 - 0.54	Distinct wave action; fluid surface; all particles resting on bed in quiescent fluid condition
Water Flood	< 0.20	< 0.41	Water flood with conventional suspended load and bedload

When routing the mud flood or mudflow over an alluvial fan or floodplain, the FLO-2D model preserves continuity for both the water and sediment. For every grid element and timestep, the change in the water and sediment volumes and the corresponding change in sediment concentration are computed. At the end of the simulation, the model reports on the amount of water and sediment removed from the study area (outflow) and the amount and location of the water and sediment remaining on the fan or in the channel (storage).

The volume of a debris detention basin upstream of the fan apex can be incorporated into the model. Small debris basins near the apex of alluvial fans may capture all or part of the mudflow event. By specifying the basin volume and grid element containing the basin in the SED.DAT file, the model will remove that portion of the hydrograph that is stored in the debris basin. If a large basin or reservoir exists, reservoir routing should be conducted prior to the application of the FLO-2D model. The reservoir outflow hydrograph can then be discretized as an inflow hydrograph to the FLO-2D grid system.

There are several important sediment concentration relationships that help to define mud and debris flows. These relationships relate the sediment concentration by volume, sediment concentration by weight, the sediment density, the mudflow mixture density and the bulking factor. When examining parameters related to mud and debris flows, it is important to identify the reported sediment concentration either by weight or by volume. The sediment concentration by volume C_v is given by:

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

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

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

where γ = specific weight of the water and γ_s = specific weight of the sediment

The specific weight of the mudflow mixture γ_m is a function of the sediment concentration by volume:

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

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

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

and

$$\rho_m = \gamma_m / g$$

where g is gravitational acceleration.

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

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

It is apparent that the bulking factor is 2.0 for sediment concentration by volume of 50%.

These basic relationships will be valuable when analyzing and reporting the results of a FLO-2D simulation. Most studies require estimates of the sediment concentration by volume and the bulking factor to describe the magnitude of the mudflow event. Average and peak sediment concentrations for the flood hydrograph are important parameters for mitigation design.

IV. MODEL CONCEPTUALIZATION

4.1 Limitations and Assumptions

Two-dimensional overland flow in FLO-2D is analyzed in a simple volume conservation, finite difference routing algorithm. It moves around blocks of fluids on a discretized flow domain consisting of a system of tiles. With appropriate estimates of flow resistance, FLO-2D numerically distributes the volume in finite fluid blocks to mimic the floodwave progression and timing over the discretized surface. Conceptually, FLO-2D is not a Lagranian fluid particle dynamics model involving streamlines or stream tubes and the movement of volume blocks around on the grid system is uniquely limited by numerical stability criteria.

The spatial and temporal resolution of the FLO-2D model is dependent on the size of the grid elements and rate of rise in the hydrograph (discharge flux). The discharge flux is distributed over the available grid element surface area for a given timestep resulting in an incremental increase or decrease in the flow depth. Finer grid resolution improves the accuracy of a flood simulation at the cost of increased computational time, more extensive data files and boundary conditions. A balance must be struck between the number of grid elements and an acceptable computational time. A grid size of 100 ft (30 m) to 1000 ft (300 m) is usually appropriate for most simulations. Smaller grid elements will not only significantly increase the number of grid elements (the number of grid elements is quadrupled each time the grid element size is divided by two), but the rate of discharge flux per unit area of the grid element increases.

FLO-2D was developed to simulate large, infrequent flood events on unconfined surfaces. The primary limitation of the FLO-2D model is the discretization of the floodplain topography into a system of square grid elements. The topography of each grid element is represented by a single elevation and roughness. Variations in topography such as mounds and depressions within the element are obscured in the model. In large flood events, variations from the average node elevation will not significantly affect the inundated area because the water surface across the element will only vary on the order of a 0.1 ft. When simulating shallow flow, steep slopes and smaller discharges, smaller grid elements should be used. Map resolution and accuracy should be considered when selecting the grid element size. Shallow flooding associated with frequent flood events may not have sufficient topographic resolution to support grid elements less than 50 ft (15 m). The loss of floodplain topographic resolution through the grid system discretization can generally be recovered in the MAPPER program when the DTM ground surface points are subtracted from the grid element water surface elevations and plotted in color contours.

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

Conceptually, the FLO-2D model distributes flow between grid elements (tiles) in a simple volume conservation scheme. By using 8 flow directions (with flow across the diagonals), the directional bias associated with 4 direction models is eliminated. Four direction models such as the DHM model require alignment of the grid elements with the primary flow direction to avoid a creating a model flow direction bias if the number of grid element rows are limited. In addition, in models that use only the diffusive wave approximation to the momentum equation, the flow will always follow the path of the steepest slope even if the flow has to turn at right angles. In this case, the artificial realignment of the grid elements may help to account for the missing convective acceleration momentum term. For this reason, an eight flow direction routing algorithm is clearly superior to one with only four directions. In an eight direction model, diagonal flow between two grid elements may crisscross, but this does not violate the principle of volume conservation.

Flood wave attenuation in the FLO-2D model is the result of simulating overbank and channel storage. This represents the interaction of the friction and bed slope terms with the diffusive pressure gradient. While the application of the dynamic wave equation can improve the flood routing, rapidly varying flow is still limited by the grid element resolution. The model does not have the ability to simulate shock waves, rapidly varying flow or hydraulic jumps, and these abrupt changes in the flow profile are smoothed out in the model's calculations. Variable subcritical and supercritical flow transitions are assimilated into the average hydraulic conditions (flow depth and velocity) between two grid elements.

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

- Steady flow for the duration of the timestep;
- Hydrostatic pressure distribution;
- Hydraulic roughness is based on steady flow resistance;
- Uniform channel geometry and roughness is represented with a channel element.

These assumptions are self-explanatory, but they help to envision that the model is simulating temporally and spatially averaged flow conditions between grid elements.

Finally, it should be noted that a rigid bed is assumed for a flood simulation, if sediment transport is not simulated. Rigid boundary conditions are appropriate for flow over steep slopes, urban flooding and mudflow events. The area of inundation associated with extreme flood events are generally unaffected by bed changes. It is assumed in rigid bed simulations that the average flow hydraulics are not appreciably affected by the scour and deposition that might occur in an individual grid element.

4.2 Finite Difference Routing Algorithms

The differential form of the continuity and momentum equations in the FLO-2D model is solved with a central, finite difference numerical integration method. The solution of the differential form of the momentum equation results from a discrete representation of the equation when applied at a single point. The finite difference numerical technique for solving the

momentum equation is an algorithm that advances the solution in time by solving for the unknown value of velocity of the dependent variables one grid element at a time. Explicit numerical schemes are simple to formulate, but are limited to small timesteps by strict numerical stability criteria. The FLO-2D requires large computational time when simulating slow rising flood waves, channels with non-prismatic features, abrupt slope change, and dead storage areas.

The solution domain for the nonlinear partial differential momentum equation is discretized into uniform square grid elements. The computational procedure for overland flow involves calculating the discharge across each of the boundaries in the eight potential flow directions. The discharge is estimated by assuming a linear trial function (central average) for depth, n-value and flow area between the two grid elements. The flow velocity across the boundary is then computed from the solution of the momentum equation. Using an average flow area between two elements, the discharge for each timestep is determined as velocity times flow area. The net change in the volume of water in each floodplain grid element for each timestep is the sum of the eight individual discharges across the boundary. Channel routing involves adding the inflow and outflow from the upstream and downstream channel elements. The channel and street routing algorithms are identical except that a rectangular channel is used for street flow.

To compute the flow velocity at a grid element boundary using the full dynamic wave equation, initially the flow velocity is calculated with the diffusive wave equation using the average water surface slope (bed slope plus pressure head gradient). This velocity is then used as a first estimate (or a seed) in the second order Newton-Raphson tangent method to find the root of the full dynamic wave equation (James, et. al., 1977). The convective acceleration and local acceleration velocities used in the solution are those determined in the previous timestep. If the Newton-Raphson solution fails to converge after 5 interations, the algorithm defaults to the diffusive wave solution. The solution algorithm incorporates the following steps where each grid element possesses a unique roughness value, elevation and flow depth:

1. The average flow geometry and roughness values between two grid elements are computed.

2. The flow depth d for computing the velocity across a grid boundary for the next timestep (i+1) is estimated from the previous timestep i using a linear trial function (the average depth between two elements).

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

3. The velocity is computed using the diffusive wave equation as previous defined in Section 2.1. The only unknown variable in the diffusive wave equation is the velocity. This is the case for overland, channel or street flow.

4. If the full dynamic wave equation is selected by the user, the predicted diffusive wave velocity for that timestep is used as a seed in the Newton-Raphson solution. It should be noted that for hyperconcentrated sediment flows such as mud and debris flows, the velocity calculations include the additional viscous and yield stress terms.

5. The discharge Q across the floodplain boundary (or between two channel elements) is computed by multiplying the velocity by the cross sectional flow area. For overland flow, the flow width is adjusted by the width reduction factors (WRFs).

6. The incremental discharge for the timestep across the eight boundaries (or upstream and downstream channel elements) are summed,

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

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

$$\Delta d_x^{i+1} = \Delta Q_x^{i+1} \Delta t \; / \; Area$$

where ΔQ_x is the net change in discharge in the eight floodplain directions for the grid element for the timestep Δt between time i and i + 1.

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

8. The simulation progresses with increasing timesteps until the stability criteria are exceeded.

In this computation sequence, the grid system inflow discharge and rainfall is computed first, then the channel flow is computed. Next, the street discharge is computed if streets are being simulated and finally, overland flow in 8-directions is determined (Figure 3). After the flow routing for each of these components has been completed, the numerical stability criteria are tested for every floodplain grid, channel or street element. If stability criteria of any of the elements is exceeded, the timestep is reduced by two times the minimum timestep and the computation sequence is restarted. If all the numerical stability criteria are successfully met, the timestep is increased by half the minimum timestep following the successful completion of the entire model computations.

For each timestep the flow depth, velocity and the discharge across every grid element boundary are calculated. The model advances explicitly by solving for the new flow depths one grid element at time. During a single sweep of the grid system for a given timestep, discharge flux is added to the inflow elements, flow velocity and discharge between grid elements are computed and the change in storage volume in each grid element for both water and sediment are determined. All the inflow volume, outflow volume, change in storage or loss from the grid system area are summed at the end of each time step and the volume conservation is computed. Results are written to the output files or to the screen at user specified output time intervals.

4.3 Parameter Sensitivity

The success and accuracy of a flood simulation can be determined in part by volume conservation. Even small data errors can result in a significant loss of volume conservation. The SUMMARY.OUT file reports on the volume conservation. It should be reviewed after each simulation to determine is the inflow volume matches the outflow volume plus storage on the grid system. While some numerical error is introduced by rounding numbers, approximations or interpolations (such as with rating tables), volume should be conservation within a fraction of cubic meter. Acceptable limits of volume conservation error for a given flood simulation is on the order of plus or minus 0.0001 to 0.000001 of one percent of the inflow.

There are no preemptive assumptions in the routing algorithm that limit the model application. It should be noted however, that the model was designed for open channel flow or steep slope overland flow applications and extensive ponding and backwater effects can required very small timesteps. For overland flow, there are two flow conditions that warrant special attention. Shallow overland flow where the flow depth is on the order of the roughness elements (>0.2 ft or 0.06 m) can be more effectively modeled by assigning the SHALLOWN parameter in the CONT.DAT file. Suggested n-values for the SHALLOWN parameter range from 0.10 to 0.20. For shallow overland flow less than 0.5 ft (0.15 m), 50% of the shallow n-value assigned in the CONT.DAT file is used. This roughness adjustment is invoked to account for higher flow resistance associated with shallow flows through vegetation. For ponded water conditions with water surface slopes less than 0.001, Manning's open channel flow equation representing the friction slope has limited applicability. Adjustments are made to the roughness n-value to reduce the velocities associated with such mild slope conditions. The adjustments increase the n-value from the original value at a slope of 0.001 to 10 times the n-value at a slope of 0.00001 based on a power regression relationship between the two slope values.

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

As was discussed in Section 3.5, limiting Froude numbers can be specified for overland flow, channel flow and street flow. When the Froude number exceeds the maximum Froude number in a grid element, the n-value is increased by 0.001 for that element for the next timestep. The flow will only exceed the limiting Froude number for that timestep. In this manner, the flow can be kept subcritical or below a specified subcritical or supercritical Froude number. For example, in steep-slope sand bed channels, high energy flows will entrain more sediment and force subcritical flow. In this case, the limiting Froude number might be set to 0.9. For flow down steep streets, a maximum Froude number of 1.2 to 1.5 may be specified to limit the supercritical flow. Since FLO-2D does not simulate hydraulic jumps, the limiting Froude number should represent average flow conditions in a channel reach. During the falling limb of

the hydrograph when the Froude decreases to a value less than 0.5, the flow resistance n-value decreases by 0.0005 until the original n-value is reached.

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

- Inflow hydrograph discharge and volume directly affect the area of inundation.
- The overland flow path is primarily a function of the topography.
- The floodplain roughness n-values generally range from 0.03 to 0.15 and control the floodwave speed over the flow domain.
- River channel roughness n-values general range from 0.020 to 0.085. N-value adjustment will usually result in only minor variation of the water surface (~ 0.2 ft or 0.06 m). The channel n-values can be used to dampen numerical surging.
- The relationship between the channel cross section flow area, bed slope and roughness controls the floodwave routing, attenuation and numerical stability. Flow area and channel element storage have the most significant effect on channel routing stability. Changes in the natural flow area between channel elements should be limited to 25% or less. More cross section surveys may be necessary to simulated rapidly changing flow geometry. Constructed rapid transitions in channel geometry can be model, but will require smaller timesteps and more channel detail.

Floodplain storage loss (ARF values) due to buildings, trees or topography can be generically assigned for the entire grid system using the XARF parameter in the CONT.DAT file. Typically, an XARF value of 5% to 10% can be applied to most floodplains. If inundation depths are relatively deep on the order of 3 ft (1 m) covering all of the grid system, the XARF can be reduced or set equal to 0.0.

Watershed and alluvial fan flooding should be bulked for sediment loading. If it is assumed that the sediment loading will be relatively minor, the XCONC factor in the CONT.DAT file can be used to uniformly bulk all the inflow hydrograph volumes. Alluvial fans that do not exhibit mudflow depositional features can be conservatively bulked using an XCONC value of 10% to 15% by volume. Typically is not necessary to bulk river flood simulation as the sediment concentrations will rarely exceed 5% by volume. Setting XCONC = 5% for river flooding will conservatively bulk the inflow hydrograph volume by 1.05. Mud and debris flow simulations should be simulated by assigning concentrations by volume to the inflow hydrographs.

4.4 Floodplain and Channel Inflow and Outflow Hydrographs

The data file for the inflow hydrographs is INFLOW.DAT. A discretized flood hydrograph from an upstream basin can be inflow either to the floodplain, channel or both. More than one grid element can have an inflow hydrograph. Hydrographs can be assigned as either direct inflow or outflow (diversions) from a channel by using the INOUTFC switch in the INFLOW.DAT file. For example, an irrigation diversion can be assigned an outflow hydrograph from any channel element. This could be a simple constant diversion of 100 cfs or a variable hydrograph over the course of the simulation. If mudflows are being simulating then a volumetric sediment concentration or sediment volume must be assigned to each water discharge increment. In an urban area, the user can specify that the inflow hydrograph be received by the floodplain element or first enter a specific street segment within the floodplain element.

Outflow control for the FLO-2D model is specified in the OUTFLOW.DAT file. For flow out of the grid system, outflow grid elements must be specified for either the floodplain or channel or both. The outflow discharge from outflow elements is set equal to sum of the inflow to that outflow element and a flow depth is then assigned to the outflow element based on a weighted average of the upstream flow depths. In this manner, normal flow is approximated at the outflow element. The outflow discharge is totally removed from the system and is accounted by the model to the outflow volume. It is possible to specify outflow from elements that are not on the boundary of the grid system, but outflow elements should be treated as sinks (all the inflow to them is lost from the flow system). Outflow elements should generally not be modified with ARF's or WRF's, levees, streets, etc. Channel outflow can also be established by a stage-discharge relationship in the OUTFLOW.DAT file. This option can be used when the channel outflow may occur at a hydraulic structure or when a known discharge relationship is available.

Stage-time relationships can be specified for either the floodplain or channel in the OUTFLOW.DAT file. These relationships can be assigned for outflow elements or for any elements in the system. When a stage-time relationship is specified, volume conservation is maintained when the discharge enters or leaves the stage-time designed grid element. Stage-time relationships provide opportunity to simulate a wide variety of coastal flood simulations related to sea storm surge, hurricane surges or tidal waves. In addition, the backwater effects of tidal variation on river and estuary flooding can be simulated. It is possible to simulate the ocean stage in urban areas without inflow hydrographs or rainfall.

4.5 Floodplain Cross Sections

A floodplain cross section analysis can be conducted by specifying grid elements in a cross section in the OUTFLOW.DAT file. The grid elements must be contiguous in a straight line to constitute a cross section across a floodplain or alluvial fan. By designating one or more cross sections, the user can track floodwave attenuation across unconfined surfaces. Both the flood hydrograph and flow hydraulics can be analyzed at cross sections. The average cross section hydraulics as well as the individual grid element hydraulics in the cross section are summarized in cross section output files, CROSSQ.OUT, HYCROSS.OUT and CROSSMAX.OUT.

4.6 Graphical User Interface and Working Environment

A graphical user interface (GUI) can be used to facilitate the data input. The GUI creates the ASCII text files used by the FLO-2D model and it requires only a few instructions to get started. To open the GUI, click on the FLO-2D Icon on your desktop or locate the FLO-2D.EXE file in the FLO-2D subdirectory. Specific instructions for the GUI are presented in the Data Input section of the manual.

The GUI is series of forms that represent the individual FLO-2D data files. Each form consists of data dialog boxes, radio switch buttons or grid entry tables. Some of the data is nested in various levels of dialog boxes that are activated as the data is entered. The dependencies of nested tables are automatically filled as the user proceeds to develop the data base from bottom to top. After the data is entered in the GUI dialog boxes, the resulting ASCII text file can be viewed from the GUI or from any other ASCII editor such as MS Word®. You can run the model or any of the processor programs from the GUI, but the model doesn't need the GUI to run a simulation. The model only requires that the data files are accessible and are constructed in the ASCII format specified in the Data Input Manual. Once the files have been created with the GUI, it may be easier to edit the FLO-2D data files with the FLOENVIR or GDS programs. Further discussion of the GUI is presented in the Data Input Manual.

A working graphical environment (FLOENVIR) can be used spatially edit the FLO-2D data base. It is possible to zoom, pan and 'zoom extents' in the program. The FLOENVIR displays the grid system and allows selection of individual or groups of grid elements using the mouse. Grid element attributes can be edited by picking individual elements or by painting a group of elements with the mouse or by drawing a polygon around a group of elements and then modifying parameters in dialog boxes. Channel, levee, streets, infiltration, area and width reduction factors, floodplain elevation and roughness, inflow and outflow nodes and rill and gully attributes can be edited. Rainfall can be spatially varied using the FLOENVIR. FLOENVIR saves the data in the ASCII format necessary to run the model. These data files can also be started from scratch after the topographical FPLAIN.DAT and CADPTS.DAT files have been created. The FLOENVIR is a particularly useful tool for editing large groups of spatial variable data such as infiltration. This processor program is discussed more fully in the second portion of the manual.

4.7 Grid Developer System (GDS)

The Grid Developer System (GDS) will generate the FLO-2D grid system from a set of digital terrain model (DTM) points. It is a pre-processor program that will overlay the grid element system on the DTM points, interpolate and assign elevations to the grid elements. It will automatically prepare the FPLAIN.DAT and CADPTS.DAT files for the FLO-2D model. The FLO-2D grid system is constructed interactively with the GDS using mouse point and click events. The data is thoroughly validated in order to minimize common input errors. The GDS supports both the English and the SI system of units.

After the GDS overlays a grid element on the limits of the DTM points, the user selects

the boundary grid elements that will constitute the outline of the FLO-2D system. The next step is to interpolate the grid element elevations from the DTM points. The user will select DTM filter criteria to interpolate the grid element elevations. Filters are available for both high and low DTM points. The need for filters arises from DTM points that may have been assigned to trees, bridges or channel beds. The GDS replaces the need for a CADD program to assign grid element elevation and significantly improves the grid element elevations, by filtering points that may distort the average floodplain elevation.

In addition to developing the FLO-2D grid The DTM also supports several important functions including the assignment of spatially variable n-value roughness, infiltration parameters, and the importation of images and GIS shape files. The GDS program incorporates ESRI MapObjects© GIS software controls. A discussion of all the various tools and components related to the ESRI and GIS interface is presented in Data Input Manual. GDS includes the following options:

- ESRI shape file format data such as land use, soil types, and Manning roughness coefficients can be imported.
- ESRI ArcInfo ASCII grid files containing terrain elevations and NOAA rainfall isopluvial data can be imported.
- Multiple geo-referenced aerial photos in various graphic formats such as TIFF, BMP, JPG, etc. can be imported and created as background.
- Multiple ESRI ArcInfo ASCII grid files can be listed in a tile and index catalog file and referenced to a user defined polygon in the study.
- Multiple image files like aerial photos can be listed in a tile and index catalog file and referenced to a user defined polygon in the study area.
- New multiple layer capability is available including extensive control of layer properties.
- Spatially variable Green-Ampt infiltration parameters can be assigned to FLO-2D grid elements based on soil shape files, land use shape files, and soil and land use properties tables.
- Spatially variable Manning roughness coefficients can be assigned to FLO-2D grid elements based on Manning shape files.
- Spatially and time variable rainfall data can be computed and assigned to FLO-2D grid elements based on multiple NOAA rain data files.
- Channel cross sections can be cut from regular or irregular elevation points and assigned to particular channel segments.

4.8 Graphical Output Options

A graphical display of the flow depths can be viewed on the screen during a FLO-2D simulation to visualize the progression of the floodwave over the potential flow surface. Different color pixels represent different flow depth increments. The display is initiated by a variable switch LGPLOT in the CONT.DAT file. The user can specify the simulation time interval for updating or refreshing the graphical display. In addition to the predicted flow depths, an inflow hydrograph will be plotted. Only one inflow hydrograph can be plotted and the user must specify which inflow channel or floodplain grid element hydrograph to view in the INFLOW.DAT file. If rainfall is simulated, the cumulative precipitation can also be plotted. If the flow depth graphics are not displayed, then a simple list of the model simulation time, minimum timestep and volume conservation are displayed on the screen. When the FLO-2D simulation is complete in the graphics mode, the maximum flow depths are displayed on the screen. If a further review of the grid element maximum depth and velocity for floodplain, channel and street flow displays are desired, a post-processor program MAXPLOT will quickly display the results including maximum and final depths and velocities. It also has a simple time series simulation whose time sequence variables are specified in TOLER.DAT.

Additional graphic programs include HYDROG, PROFILES and MAPPER. HYDROG is a post-processor program that will plot the hydrograph for every channel element. The user can quickly move up and down the channel, viewing the channel hydrograph out of the channel element. The hydrographs can be printed or saved as an ASCII data file or *.bmp image. HYDROG can also be used to evaluate the average channel hydraulics in a given reach. The user can select the upstream and downstream channel elements and the program will compute the average of the hydraulics for all the channel elements in the reach including: velocity, depth, discharge, flow area, hydraulic radius, wetted perimeter, top width, width to depth ratio, energy slope, and bed shear stress. An optional data file of measured hydrographs (HYDRO.DAT) can be prepared for comparison with the FLO-2D predicted hydrograph. These two hydrographs are plotted together.

The PROFILES program plots channel water surface and bed slopes. It also displays survey cross section data in a profile plot and table so the channel cross section geometry can be edited and interpolated. PROFILES is an important useful tool when building and editing the channel data file. Generally, one cross section is surveyed for every 4 to 5 channel grid elements. It is necessary therefore to interpolate cross sections for channel elements without surveyed cross sections. This can be accomplished by simply selecting the upstream and downstream channel elements and clicking the interpolation button. Flow area weighted cross sections will be distributed to those channel element between the know cross sections. Both cross section shape and slope are interpolated.

MAPPER is the primary program for displaying and utilized the FLO-2D flood simulation results. It can be used to develop production color contour plots of the FLO-2D results either on a grid element or DTM point basis including hydraulic variables and water surface elevations. Three types of maps can be created: grid element or DTM point plots, contour maps and shaded contour maps. These graphic maps may be created for ground surface elevations, maximum water surface elevations, maximum floodplain flow depths and maximum

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

MAPPER version 3.0 has the following features:

- Incorporates ESRI MapObjects© GIS software controls.
- ESRI shape file format data such as land use, soil types, and Manning roughness coefficients can be imported.
- ESRI ArcInfo ASCII grid files containing terrain elevations can be imported.
- Multiple geo-referenced aerial photos in various graphic formats can be imported such as TIFF, BMP, JPG, etc.
- Multiple ESRI ArcInfo ASCII grid files listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- Multiple image files such as aerial photos listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- New multiple layer capability, including control of layer properties is available.

MAPPER can also export various types of graphic files for use in other CADD applications. GIS shape files (*.shp) are automatically created with any plotted results. This GIS shape files can be then be imported into ArcView or other GIS programs. All of these post-processor programs are described in detail in the input data section of the manual. The MAPPER tool will continue to be expanded in the future. FEMA FIS flood map templates are being prepared for the next version of FLO-2D.

4.9 Data Output Options

The FLO-2D model has a number of output options and files to help the user organize the results. Floodplain, channel and street hydraulics are written to file. Hydraulic data include water surface elevation, flow depth and velocities in the eight flow directions. Discharge and sediment concentration for specified output intervals (hydrographs) are written to various files. A mass conservation summary table comparing the inflow, outflow and storage in the system is presented in the SUMMARY.OUT file and at the end of the BASE.OUT file. Several options available to format the output files to generate either temporally or spatially varied results. Some output files are created by simply initiating the various flow components (e.g. STREET.OUT is created when street flow is simulated). A complete description of the output files are presented in the Data Input Manual. Output files that are generated by FLO-2D include:

BASE.OUT - This output file lists the inflow hydrograph, flow hydraulics for each output timestep, summary listing of the maximum flow hydraulics and summary mass conservation

table. All the hydraulic output data for every grid element and timestep can be written to this file. Use the NOPRTFP and NOPRTC variable switches in the CONT.DAT to limit the output in this file.

CHANGRID.OUT - The channel elements that experience significant gains or losses of flow volume are listed in this file. These channel elements can be reviewed for volume conservation problems.

CHANMAX.OUT - This output file lists the peak discharge for each channel element along with its corresponding time of occurrence.

CHANBANKEL.CHK - When the bank elevations prescribed in the CHAN.DAT do not match the corresponding floodplain grid element elevations, the difference in elevation is reported in this file. If the difference between the channel bank elevation and the floodplain grid element elevation exceeds 1 ft (0.3 m), the model reports it in this file. This includes the channel extension grid elements. If the 1 ft (0.3 m) limit is exceeded during the simulation, the model assumes that the bank elevation is a more accurate representation of the floodplain for that grid element and resets the floodplain elevation to the bank elevation.

CHANNEL.CHK - The grid elements which encompass a channel cross section are listed in this file. Review this file when the channel is wider than the width of a grid element.

CHANWS.OUT - This output file contains a list of channel element, x- and y-coordinate and channel water surface elevation.

CHNBEDEL.OUT - This file is a list of the channel element bed elevations used in the MAXPLOT post-processor program.

CHVOLUME.OUT - Presented in this file is a summary of the channel volume conservation including channel inflow, outflow, overbank flow, return flow from the floodplain, infiltration losses and channel storage. Review this file to discern if the channel volume is being conserved.

CONFLUENCE.OUT - In this file there is a list of channel elements that constitute a confluence as defined by having three or more channel elements contiguous to a given channel element. The confluence may be a prescribed condition or it could constitute a data error in the CHAN.DAT requiring NOFLOC definition.

CROSSQ.OUT - The discharge hydrograph for the cross section grid elements listed in the OUTFLOW.OUT file are presented. The floodplain cross section option must be initiated.

CROSSMAX.OUT - This is an output file created with the floodplain cross section option that will list the peak discharge for each floodplain cross section by grid element and time of occurrence.

DEPTH.OUT - This file contains the predicted maximum flow depths for either floodplain, street or channel grid element whichever is the greatest. It lists the maximum flow depth for each grid

element together with the x- and y-coordinate in a format that can be imported to CADD software. The CADD program can then be used to create maximum flow depth contours using the digital terrain model. Related maximum depth output files with grid element number, coordinates and depth include:

DEPCH.OUT - Maximum channel flow depths.
DEPFP.OUT - Maximum floodplain flow depths.
DEPTH.OUT - Maximum combined channel, street and floodplain flow depths.
FINALDEP.OUT - Final floodplain flow depths.

DEPTHDUR.OUT - The flood duration for a prescribed flow depth is written to this file along with grid element numbers and coordinates. The user specifies a floodplain flow depth and then the model tracks the time duration that the grid element is flooded above prescribed flow depth.

DEPTHTOL.OUT - This file contains the maximum combined channel and floodplain flow depths greater than the TOL value. Values less than the TOL value are set to zero.

ERROR.CHK - Check this file for data input errors when the program prematurely terminates. The backup files (*.BAC) should be reviewed in conjunction with this file.

FPSUPER.OUT - When the overland flow is supercritical, the grid elements, time of occurrence and Froude Number are written to this file.

HYCHAN.OUT - This output file lists the channel flow hydraulics including elevation, flow depth, velocity, discharge and sediment discharge by grid element in hydrograph form. It also lists the maximum discharge, stage and their time of occurrence. Additional channel flow hydraulic and geometry parameters include: flow area, hydraulic radius, wetted perimeter, top width, width-to-depth ratio, energy slope, and bed shear stress.

HYCROSS.OUT - This file lists all the floodplain cross section hydraulic data including elevation, top width, flow depth, and velocity, discharge and sediment concentration. The floodplain cross section option must be initiated.

HYDROSTRUCT.OUT - The discharge hydrograph from any hydraulic structure listed in the HYDROSTRUC.DAT file is presented in this output file.

IMPACT.OUT - This file list the impact force per linear foot of structure on the floodplain. The data is presented by grid element number, coordinates and force per linear foot.

INFILHY.OUT - The hydraulic conductivities are listed in this file to view the spatial variation. This file contains grid element number, coordinates and floodplain hydraulic conductivity.

INTERGWS.OUT - The maximum floodplain water surface elevations based on the grid element elevation in the TOPO.ORI are presented in this file. Values less than TOL are set to zero.

LEVEE.OUT - The LEVEE.OUT file contains a list of the grid elements with a failed or overtopped levee. If the levee has failed, the failure width, levee elevation, discharge and the time of failure are listed. If the levee has only been overtopped without failure a output interval time and discharge are presented.

MULTCHN.OUT - Adjustments to the multiple channel widths during the model simulation are listed in this file. These adjustments are dependent on the gully bankfull discharge and incremental width increases.

OUTNQ.OUT - Hydrographs for the floodplain outflow grid elements are listed in this file.

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.

ROUGH.OUT - This file can be reviewed for maximum n-values computed when the predicted Froude number exceeds the specified limiting Froude numbers. The final and original n-values are also written to this file.

SEDCHAN.OUT - This file lists the maximum changes in the channel bed elevations resulting from the application of the sediment transport routing algorithm. The x- and y-coordinates, maximum deposition, maximum scour and final elevation changes by grid element are written to this file.

SEDFP.OUT - This file is the companion output file to SEDCHAN.OUT file for floodplain elements. The maximum changes in the bed elevation for each floodplain grid element are recorded.

SEDTRAN.OUT - The sediment transport capacity (cfs or cms) computations for each of the six sediment transport equations are listed by output interval in this file.

SPECENERGY.OUT – The specific energy is the sum of the depth plus the velocity head. This file lists the maximum specific energy for a floodplain grid element and includes grid element number, coordinates and maximum specific energy.

STATICPRESS.OUT - The spatially variable static force per linear foot for each floodplain element is presented is this file by grid element number, coordinates and force per linear foot.

STREET.OUT - This file is similar to the DEPTH.OUT file. It contains the grid element number, x- and y-coordinates and the predicted maximum street flow depths.

STREVEL.OUT - This file is a list of the street elevations used in the MAXPLOT post processor program.

SUMMARY.OUT - The mass conservation table comparing inflow, outflow and storage is presented in this output file. This table is also listed at the end of the BASE.OUT file.

SUPER.OUT - When the floodplain or channel flow is supercritical, the grid elements experiencing supercritical flow, Froude number, discharge and time of occurrence are written to this file. This file requires that the variable SUPER = 1 in the CONT.DAT file.

SURFAREA.OUT - This file summarizes all the available surface storage area by grid element in the system including the channel surface area, street surface area, multiple channel area, floodplain areas reduced with the area reduction factors and the remaining floodplain surface area available for flow storage. At the end of the file is an output interval listing of the wetted area of inundation by floodplain, channel and combined maximum. This output will indicate approximately when the maximum surface area was inundated.

TIME.OUT - The twenty grid elements associated with the most timestep decreases are listed in this file. This file identifies which floodplain, channel or street elements are the most frequent in slowing down the model. These are the grid elements whose attributes the user can modify to speed up the flood simulation.

TIMDEP.OUT - The predicted flow depths are written to this file according to grid element together with the x- and y-coordinates for prescribed output intervals for a time lapse display of the flood progression. This file can be imported to the CADD drawing to create time lapse flow contours or a 3-dimensional display of the flood movement. MAXPLOT will create an animation of the flood event using this file.

VELOC.OUT - Similar to the DEPTH.OUT file, this file contains the predicted maximum velocity for each grid element along with the x- and y-coordinates. Maximum velocity contours can be plotted in MAXPLOT, MAPPER or in CADD software. Related maximum velocity files with grid element number, coordinates and velocity data include:

FINALVEL.OUT - Final overland flow velocities at the end of the simulation
FINALVELDIR.OUT- Final maximum overland flow velocity components at the end of the simulation.
STVEL.OUT - Maximum street flow velocity.
STVELDIR.OUT - Flow direction of the maximum street flow velocity component.
VELFP.OUT - Maximum floodplain flow velocity
VELOC.OUT - Maximum combined channel, street and floodplain flow velocity.
VELDIREC.OUT - Flow direction of the maximum floodplain flow velocity component.

XSECAREA.OUT - This output file contains cross section geometry data including bankfull flow area, wetted perimeter and top width.

XSEC.OUT - This file is created by the sediment transport option (ISED = 1 in CONT.DAT) with channel cross section data. It contains the final cross section bed elevations after scour and deposition have been computed in a mobile bed simulation.

The MAXPLOT program uses the following FLO-2D output files for a graphical display of the maximum depths and velocities (these files contain grid element number, x- and y-coordinates, and depth, velocity or elevation):

CHNBEDEL.OUT - Channel element bed elevations.
DEPCH.OUT - Maximum channel depths.
DEPTHDUR.OUT - Duration of flooding about the specified depth in hours.
DEPFP.OUT - Maximum floodplain depths.
DEPTH.OUT - Maximum combined channel, street and floodplain flow depths.
FINALDEP.OUT - The final channel or floodplain depths at the end of the simulation.
FINALVEL.OUT - Final velocity at the end of the simulation.
FINALDIR.OUT - Final flow velocity direction at the end of the simulation.
SEDFP.OUT - Maximum changes in the grid element bed elevation.
STREET.OUT - Maximum street flow depths.
STREVEL.OUT - Street elevations.
STVEL.OUT - Maximum street flow velocities.
STVELDIR.OUT - Flow direction of the maximum street flow velocity components.
VELDIRECT.OUT - The maximum flow velocity directions.
VELFP.OUT - The maximum flow velocities.
VELOC.OUT - Maximum combined channel, street and floodplain flow velocities.

4.10 Starting the Program

After the data files have been constructed with the GUI or with an ASCII editor, the model is ready to run. Only six data files are required to run an overland flow simulation, FPLAIN.DAT, CADPTS.DAT, CONT.DAT, TOLER.DAT, INFLOW.DAT and OUTFLOW.DAT. From Windows Explorer, open the project subdirectory containing the data files, make sure that the FLO.EXE is in the subdirectory and double-click on FLO.EXE to start a flood simulation. No other commands are necessary. Most users will open the FLO-2D GUI by clicking on the desktop icon and then activate the FLO-2D model using the 'Execute' button from the pull down menu to start a flood simulation. All the processor programs are accessible from the GUI pull down menus. In addition, any of the processor programs can be run by putting the *.EXE file for the processor into the project subdirectory and double-clicking on the processor name.

4.11 Code Language and Hardware Requirements

The FLO-2D model software is written in Lahey Fortran 95 computer language. Windows-based graphics code and subroutines have been written in the Winteracter 3.0 language from Interactive Software Services, Ltd. of the United Kingdom. The FLO-2D GUI, GDS and MAPPER processor programs were written in Visual Basic. There are no hardware limitations for computers less than two years old with a clock speed greater than 1.5 MHz and either a Windows 98, 2000, XP or NT operating systems. Almost all of the variable arrays are allocated at model runtime so the number of grid elements and associated components such as channels and streets are essentially unlimited. On occasion an older computer may have a DOS default to an integer array limitation of 32,767. This means that the number of grids should be less than 32,000. The normal integer array size default is 2,147,483,647. A complete list of data array size limitations and variable range is provided in the second portion of the FLO-2D manual.

V. FLO-2D APPLICATIONS AND METHODS

5.1 River Applications

Simulating channel flow is one more difficult facets of applying the FLO-2D model. Abrupt channel cross section transitions, flat bed slopes, confluences and limited data bases are some of the difficulties that have to be addressed. The key to effective FLO-2D river flood applications is correctly assessing the relationship between the volume stored in the channel and the volume distributed on the floodplain. This requires good definition of the relationship between channel flow area, slope and roughness. There are several considerations to defining channel volume and geometry. An accurate estimate of the total channel length (sum of the channel grid element lengths) is critical to channel volume computation. In addition, the surveyed channel cross sections should be appropriately spaced to adequately model transitions between wide and narrow cross sections. Finally, bankfull water surface elevations should be surveyed at cross sections to calibrate the channel roughness values. Modeling channel geometry with a limited data base can result in discharge surging, which can be observed in a plot of the channel element hydrographs. The consequence of an inadequate channel data base is small timesteps and long flood simulations to reduce or eliminate the numerical surging.

Once the FLO-2D topography files FPLAIN.DAT and CADPTS.DAT have been created, the channel system can be laid out. This is easily accomplished in the FLOENVIR program by first selecting channel grid elements and then assigning the various channel attributes. After the CHAN.DAT file has been created, the channel bed slope and cross sections can be adjusted in the PROFILES program. The program displays the channel profile and the cross section plots which can be edited directly in PROFILES. When one cross section has been assigned to several grid elements, the bed slope through this reach will be flat. Between channel elements with surveyed cross sections, the bed slope can be interpolated in the PROFILES program to create a uniform bed slope. Similarly, between channel elements with surveyed cross sections, the assigned cross sections can be interpolated to create a smooth transition between wide and narrow surveyed cross sections.

When preparing a channel simulation, the available cross section data is distributed to the various channel elements based on the top width observed on mapping, images or aerial photos. The bed elevation is then adjusted between cross sections to achieve a uniform slope between the surveyed cross sections. The n-value assignment is estimated from either calibration of surveyed water surface elevations or from knowledge of the bed material, bed forms, vegetation or channel planform. The n-values also serve to correct any mismatched channel flow area and slope. By specifying a maximum Froude number that represents bankfull flow conditions for a given reach, the model will increase the n-values gradually when the limiting Froude number is exceeded. The maximum n-value can be reviewed in the ROUGH.OUT file and then the n-values in the channel can be adjusted. If the n-value adjustment appears to be outside an acceptable range, the flow area or the bed slope can be revised. Using this approach, the relationship between the channel flow area, bed slope and n-value can be adjusted to better represent the physical system, calibrate the water surface elevations, and reduce surging and speed-up the simulation. The limiting Froude will also keep the average velocity in an acceptable range of the wave celerity.

The floodplain inundation from channel overbank flow is a function of the channel conveyance and the floodplain roughness and topography. To improve the accuracy of a flood inundation analysis, the channel conveyance and water surface elevations should be calibrated by modifying the n-values, the flow area or the bed slope. The floodplain topography can then be checked to insure that the grid element elevations are appropriate. The speed of the floodwave progression over the floodplain can be adjusted with the floodplain n-values.

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

Flooding routing details for channel applications include simulating hydraulic structures, levees, infiltration, sediment transport and hyperconcentrated sediment flows. Hydraulic structure rating curves and rating tables are input in the HYDROSTRUCT.DAT file and may include bridges, culverts, weirs, diversions or any other channel hydraulic control. Levees are usually setback from the river on the floodplain, but can control the water surface in the channel if the flood is confined by the floodplain levees. Channel infiltration is based solely on the hydraulic conductivity in the INFIL.DAT file and represents average bed and bank seepage conditions. Bed scour or deposition associated with a mobile analysis is nonuniformly distributed on the channel cross section. Finally, mudflows can be routed in channels.

5.2 Overland Flow and Alluvial Fan Applications

An unconfined flood simulation on a floodplain or alluvial fan without channels will usually have larger timesteps and be more numerically stable than channel flood simulations because the flood volume is distributed over a large surface area. A water flood simulation over a floodplain or alluvial fan surface requires only four additional files after topography files (FPLAIN.DAT and CADPTS.DAT) have been created; CONT.DAT, TOLER.DAT, INFLOW.DAT and OUTFLOW.DAT. The CONT.DAT and TOLER.DAT files have just a few lines of control and numerical stability data. The INFLOW.DAT file contains the inflow hydrograph and the OUTFLOW.DAT lists the outflow nodes. If the flood hydrology is available, FLO-2D floodplain simulation without channels can be running within minutes after the topography files are created.

The primary focus of an unconfined flood simulation is based on those factors that control volume distribution over the surface. The important flood routing details are accurate topography, spatial variation in infiltration and roughness and features such as flow obstructions and streets. Street flow is important for distributing shallow flooding in urban areas. Buildings and walls that obstruct flow paths and or eliminate floodplain storage are important for flow path redirection. Other flood routing details include levees, hydraulic conveyance facilities such as culverts, rainfall and gully flow. The levee routine can be used to simulate berms, elevated road fill, railroad embankments or other topographic features.

It is possible to use the overland flow components to model the interior of various floodplain features such as detention basins, river channels or even streets. Flood retention basins have been modeled as part of the entire floodplain system using either the grid element elevation or levees to define the basin storage area. An appropriate grid element size should be selected to generate enough interior elements to adequately simulate the basin or channel. It should be noted that modeling the channel interior may require very small timesteps. Manning's steady flow, open channel flow equation for the friction slope may not be appropriate if the ponded water surface is very flat.

FLO-2D can simulate an unconfined overland floodwave progression over a dry flow domain without any specifying any boundary criteria. No hot starts or prescribed water surface elevations are required. There is no limit to the number of inflow hydrograph elements. Outflow nodes must be designated but required no specific boundary conditions or water surface control. Outflow from the grid system is approximate as normal depth flow using a weighted average flow condition of the contiguous upstream grid elements.

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

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

5.3 Project Results – What is a Successful Flood Simulation?

When a FLO-2D simulation is completed, how do you that the simulation was successful or accurate? Where should you look first to determine if the results are meaningful? The first output file that should be reviewed is the SUMMARY.OUT file. In this output file, volume conservation results and the ultimate disposition of all the inflow hydrographs and rainfall can be examined. If the volume was not conserved, then it will be necessary to conduct a more detailed review of the SUMMARY.OUT and other files to determine when the volume error was initiated

and whether the channel volume was conserved (CHVOLUME.OUT). If the volume was conserved, then the area of inundation can be quickly reviewed in either MAXPLOT or MAPPER programs. The area of inundation may be limited if the channel elements were disconnected or if a routing component did not have correct data. If the area of inundation seems reasonable and the flood appears to have progressed completely through the system, then the channel flow should be reviewed for surging in either CHANMAX.OUT of by scanning the channel element hydrographs in the HYDROG program.

Once the FLO-2D flood simulation is providing good results, the next step is to fine tune the model and add more detail. To speed up the model, review the TIME.OUT file to determine which channel floodplain or street elements are causing the most timestep reductions. If the number of timesteps reductions for a grid element is an order of magnitude or greater than the rest of the elements, its topography, channel geometry, slope or roughness in relationship to its neighbor grid elements can be adjusted. Model speed is not critical if the simulation is accurate with respect to volume conservation, discharge surging and area of inundation unless a number of project scenario simulations are required.

During the review process, the ROUGH.OUT file can provide insight into the model analysis. Channel, street or floodplain elements with large increases in roughness in response to reasonable limiting Froude numbers should be reviewed and adjusted. High Froude numbers may be the result of discharge surging and this can be initially addressed with more appropriate or spatially varied n-values. It may necessary to increase the n-values for several elements in a given reach or area. If the model does not respond to higher n-values, adjusting the slope or flow area may be required.

For mudflow flow simulations, the sediment volume reported in the SUMMARY.OUT file should be reviewed. Very viscous mudflows should have an average concentration by volume in the range from 25% to 35%. The total inflow sediment volume should represent the potential sediment yield from the watershed for the simulated return period volume. A rough check of the potential sediment supply should be reviewed including hillslope failure, bank erosion, channel scour and overland sediment yield.

In the Data Input portion of the manual, further discussion of model analysis and trouble shooting is presented. When it seem unclear as to where the project model is not functioning correctly, simplify the simulation by turning off components. For example, turn off all the peripheral model details except for the channel and the floodplain. Make certain that these two components are providing accurate results. Then add components back to the model, one component for each model simulation, turning off the previous reviewed components. In this manner, the model component that may be causing the problem will be revealed. Reducing the timesteps should never be overlooked when addressing issues such as discharge surging and erratic results.

VI. FLO-2D MODEL VERIFICATION

6.1. General

The FLO-2D model has been applied on countless projects by federal and state agencies, consulting firms and international users. Most users perform some sort of verification test to convince themselves of the validity of the flow routing algorithms. A Swiss research organization conducted a flume test to compare measured data with model predicted hydraulics. All users have to address the concern of whether the FLO-2D predicted hydraulics are reasonable and accurate for their projects. On occasion, an application may require some code modification to address a unique flow problem.

In January 1999, the Sacramento District Corps of Engineers accepted of the FLO-2D model for overland flow and alluvial fan flood studies. They conducted a review of several model test applications, applied the model to a number of unconfined flood hazard projects, developed the California Aqueduct test case and replicated of the Arroyo Pasajero March 1995 alluvial fan flooding. Over a three-year period, the Corps actively engaged in model enhancement, code modification and model testing to expand the model applicability. In early 2001, the Albuquerque District of the Corps of Engineers completed a review of the FLO-2D model for riverine studies. Much of the supporting documentation and application test cases are described in this section. Both the Sacramento and Albuquerque District submitted acceptance letters to FEMA in support of using the FLO-2D for flood insurance studies. The FLO-2D model is on FEMA's list of approved hydraulic models for both riverine and overland flow (alluvial fan) flood studies.

Verification of hydraulic models with actual flood events is dependent on several factors including estimates of flow volume and area of inundation, appropriate estimates of flow resistance, representative conveyance geometry, accurate overland topography and measured flow hydraulics including water surface elevation, velocities and flow depths. Ideally, the best model test involves the prediction of a flood event before it occurred; however, the probability of an actual flood having the exact volume of the predicted flood event is remote. Verification Test 4, the Green River routing model provides an opportunity to test the model with actual flow releases from a dam. The tools for verifying hydraulic models include physical model (prototype) studies, comparison with other hydraulic numerical models or replication of past flood events. To review the accuracy of the FLO-2D model all three verification methods are employed. The primary verification issues are the accuracy of the channel and the unconfined overland flow routing algorithms and the area of inundation.

A series of application tests are presented for the FLO-2D model verification:

- Channel flow using the full dynamic wave equation on the mild sloped California Aqueduct;
- Channel flow routing component compared to HEC-2 model results;
- Channel and floodplain flow routing for an actual river flood, Truckee River, 1997;
- Channel routing in a major river system (Green River) with prediction of a dam release floodwave movement;
- Comparison of floodplain inundation with mapped wetted acreage (Middle Rio Grande);
- Verification of mudflow hydraulics.

The results of these tests demonstrate that the FLO-2D computation algorithms are accurate for both channel and overland flood routing. The results also demonstrate the importance of routing the entire flow hydrograph. All the verification tests for Albuquerque District review were rerun with the most recent version of the FLO-2D model.

6.2. Verification Test 1. Channel Hydraulics in the California Aqueduct

Discharge in a 10 mile reach of the California Aqueduct near Coalinga, California was simulated using 96 channel elements in a 500 ft square grid system. The aqueduct is essentially a large outdoor physical model and is one of the most difficult hydraulic simulations for a finite difference model because of the high discharge flux, very low Froude number flow and very mild slope (5.7×10^{-5}). Typically numerical surging is difficult to control in such cases. This test application is comparable to modeling flow in the Mississippi River.

The design discharge for this concrete lined trapezoidal channel was approximately 8,150 cfs. The roughness n-values for most of the 10 mile reach ranged from 0.020 to 0.024 with a few grid elements having slightly higher n-values of 0.025 or 0.026. The variability in roughness accounted for the irregularities in the canal bed due to subsidence in the Central Valley resulting from long term groundwater extraction. The simulation time was 7 hours to create uniform flow throughout the reach for a constant discharge of 8,150 cfs. At the end of the simulation, the inflow discharge was 8,150 cfs and the outflow discharge 8,149 cfs. Hydraulic design conditions for the canal reach were an average flow depth of approximately 26 ft and an average velocity of 3 fps. The prescribed operating water surface elevations were 317.26 at the start of the reach and 314.26 at the end of the reach. The FLO-2D predicted water surface elevations were 317.15 and 314.16 for the inflow and outflow nodes.

The results are presented in Table 10 and show consistent discharge throughout the reach using the full dynamic wave momentum equation. The predicted velocities and flow depths are very uniform and precisely replicate the hydraulic design conditions. This application verified the FLO-2D dynamic wave routing algorithm for a large channel with a very mild slope.

Inflow discharge: 8150.00 cfs

NODE	ELEV.	DEPTH	VELOCITY	Q (cfs)
3865	317.15	26.89	2.68	8151.98
3960	316.92	26.70	2.69	8149.34
4055	316.89	26.71	2.69	8148.58
4150	316.86	26.69	2.69	8150.32
4245	316.83	26.67	2.70	8151.07
4340	316.81	26.66	2.70	8150.21
4436	316.78	26.64	2.70	8149.71
4532	316.75	26.62	2.71	8149.55
4531	316.72	26.59	2.71	8149.73
4530	316.70	26.58	2.71	8150.50
4626	316.67	26.53	2.72	8151.57
4721	316.64	26.48	2.73	8150.87
4720	316.62	26.44	2.73	8149.20
4719	316.59	26.39	2.74	8148.06
4815	316.56	26.34	2.75	8149.64
4911	316.53	26.29	2.76	8151.30
4910	316.51	26.25	2.76	8151.52
4909	316.48	26.21	2.77	8150.13
4998	316.45	26.15	2.78	8147.99
5086	316.41	26.09	2.79	8148.02
5085	316.38	26.04	2.80	8149.15
5084	316.35	25.97	2.80	8150.63
5171	316.31	25.96	2.80	8151.02
5258	316.28	26.00	2.93	8150.10
5257	316.23	26.01	3.07	8149.12
5256	316.17	26.03	3.07	8148.72
5342	316.13	26.05	3.06	8149.26
5341	316.09	26.13	3.05	8149.85
5340	316.06	26.23	3.03	8150.00
5425	316.02	26.35	3.01	8149.32
5424	315.99	26.44	3.00	8148.53
5423	315.95	26.45	2.99	8147.97
5507	315.92	26.45	2.99	8148.20
5506	315.89	26.44	3.00	8148.83
5505	315.86	26.42	3.00	8149.61
5587	315.82	26.41	3.00	8149.95
5586	315.79	26.40	3.00	8149.77
5585	315.76	26.39	3.00	8149.26
5584	315.73	26.39	3.01	8148.56
5664	315.70	26.37	3.01	8148.11
5663	315.67	26.36	3.01	8148.05
5662	315.64	26.35	3.01	8148.26
5661	315.61	26.35	3.01	8148.68
5738	315.58	26.37	3.01	8149.02
5737	315.56	26.39	3.01	8149.21
5736	315.53	26.39	3.00	8149.30
5810	315.50	26.41	3.00	8149.35
5809	315.47	26.42	3.00	8149.43
5808	315.44	26.42	3.00	8149.53
5807	315.41	26.43	3.00	8149.60
5806	315.37	26.42	3.00	8149.56
5805	315.34	26.43	3.00	8149.31
5804	315.30	26.40	3.01	8148.84
5803	315.27	26.37	3.01	8148.23
5802	315.23	26.36	3.01	8147.66
5801	315.20	26.37	3.01	8147.33
5800	315.17	26.39	3.00	8147.37
5799	315.14	26.40	3.00	8147.75
5798	315.11	26.41	3.00	8148.30
5797	315.08	26.42	3.00	8148.79
5796	315.06	26.44	3.00	8149.06
5795	315.03	26.45	2.99	8149.04
5794	315.00	26.46	2.99	8148.81
5793	314.98	26.49	2.99	8148.52
5792	314.95	26.50	2.98	8148.28
5791	314.93	26.52	2.98	8148.17
5790	314.90	26.53	2.98	8148.17
5789	314.88	26.55	2.98	8148.22
5788	314.86	26.57	2.97	8148.25
5787	314.83	26.59	2.97	8148.23
5786	314.81	26.61	2.97	8148.16
5785	314.78	26.62	2.96	8148.09
5784	314.76	26.64	2.96	8148.05
5783	314.73	26.65	2.96	8148.06
5782	314.71	26.64	2.96	8148.11
5706	314.68	26.64	2.96	8148.17
5627	314.65	26.64	2.96	8148.20
5626	314.62	26.63	2.96	8148.16
5544	314.59	26.63	2.96	8148.07
5543	314.57	26.63	2.96	8147.97
5459	314.54	26.63	2.96	8147.90
5458	314.51	26.62	2.96	8147.89
5372	314.48	26.62	2.96	8147.99
5285	314.45	26.63	2.96	8148.12
5284	314.43	26.63	2.96	8148.21
5283	314.40	26.63	2.96	8148.27
5282	314.38	26.63	2.96	8148.28
5281	314.35	26.62	2.97	8148.29
5280	314.33	26.62	2.97	8148.29
5279	314.30	26.61	2.97	8148.28
5191	314.27	26.61	2.97	8148.20
5102	314.24	26.61	2.97	8148.04
5101	314.21	26.60	2.97	8147.88
5012	314.19	26.61	2.97	8147.79
5011	314.16	26.60	2.97	8149.01

Table 10. California Aqueduct Predicted Hydraulics

6.3. Verification Test 2. Variable Channel Geometry Hydraulics

This simulation test was performed in 1989 to verify the channel routing component by comparing the results with the Corps of Engineers HEC-2 hydraulic model. The purpose of this test was to demonstrate FLO-2D's capability to replicate HEC-2 flow hydraulics using an identical cross section data base and roughness values. The initial FLO-2D tests were conducted using only the diffusive wave approximation to the full momentum equation. The simulation test is reproduced for this document using the full dynamic wave equation to compute the flow velocities.

Most hydraulic model testing involves the replication of project field data through the calibration of roughness parameters. Then to complete the test, the calibrated model may be applied to another field data base. A second, more stringent test compares the hydraulic computations from another accepted hydraulic model using identical data sets. Although the FLO-2D model is a flood routing model and HEC-2 predicts hydraulics using a step backwater calculation for one discharge at a time, the model comparison was accomplished by running FLO-2D with the same discharge for a sufficiently long simulation time to achieve uniform, steady flow throughout the entire reach.

A hypothetical channel 22,500 feet long with 15 variable area geometry cross sections and four different slope reaches, two steep and two mild reaches was created (Figure 10). Actual river cross sections were used in the test, but the roughness and slope were assigned fictitious values to create supercritical and subcritical flows in alternate reaches. The HEC-2 model was run twice for each discharge, to analyze both the subcritical and supercritical flow regimes. For the FLO-2D simulation, the channel elements were 500 ft in length. It should be noted that the cross-sectional geometry did not necessarily correlate with the fictitious slope and roughness values and that the assumed associations between cross section, slope and roughness would not occur naturally.

In the initial tests in 1989, discharges ranging from 100 cfs to 10,000 were simulated with similar results. Representative results from the 5,000 cfs simulation are presented in Figures 11 and 12. As shown in these figures, FLO-2D predicted flow depths and velocities that compare well with the HEC-2 hydraulics. The differences in the results of the two models arise from interpreting the HEC-2 hydraulics in the slope transitions as either subcritical or supercritical flow. FLO-2D predicted average transitional flow hydraulics between subcritical and supercritical regimes and any hydraulic jumps are lost in the computation between 500 ft grid elements. There is the added difficulty of interpolating the results at a given reach station since the HEC-2 cross sections are not necessarily located at grid element node. Given the contrived channel geometry with artificial slope and roughness, the results show that natural channel routing can be conduced with FLO-2D. This test also demonstrates the consistency of the FLO-2D model as it evolved over the years.

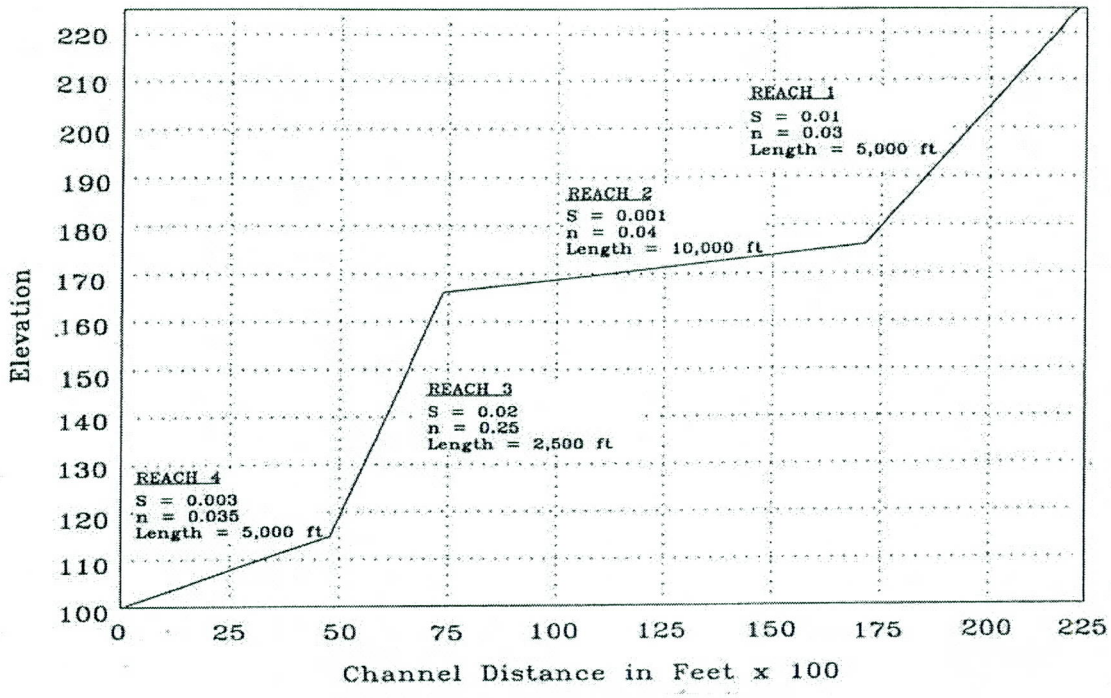


Figure 16. Channel Profile

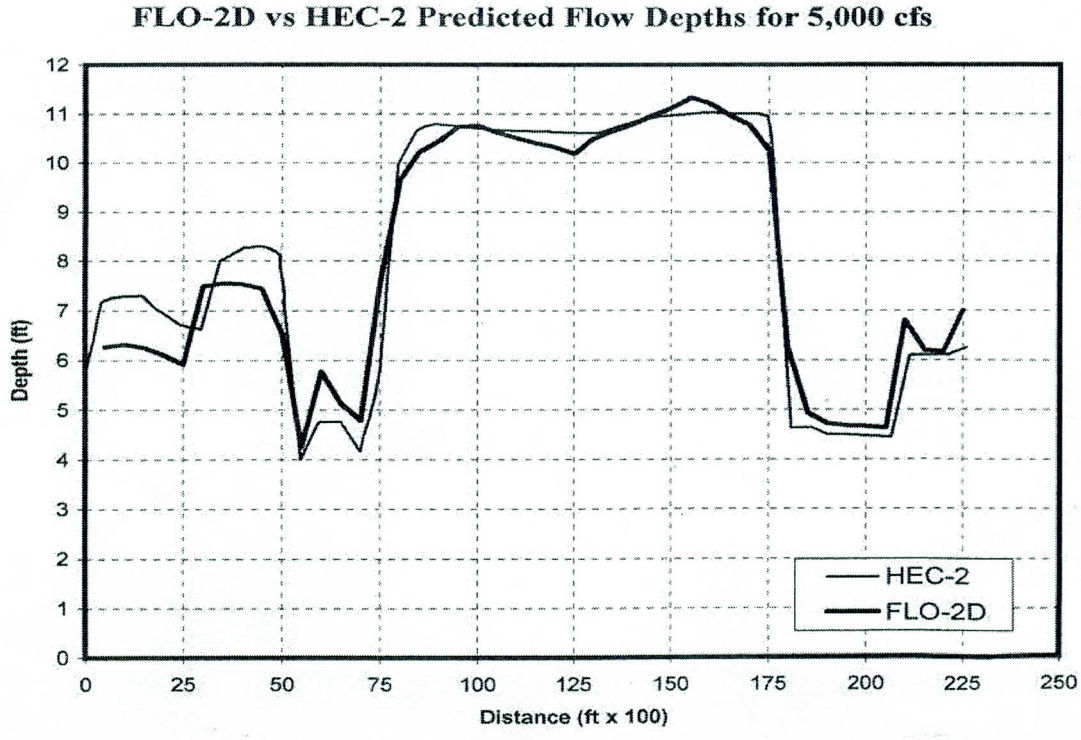


Figure 17. Comparison of FLO-2D and HEC-2 Flow Depths for 5,000 cfs Discharge

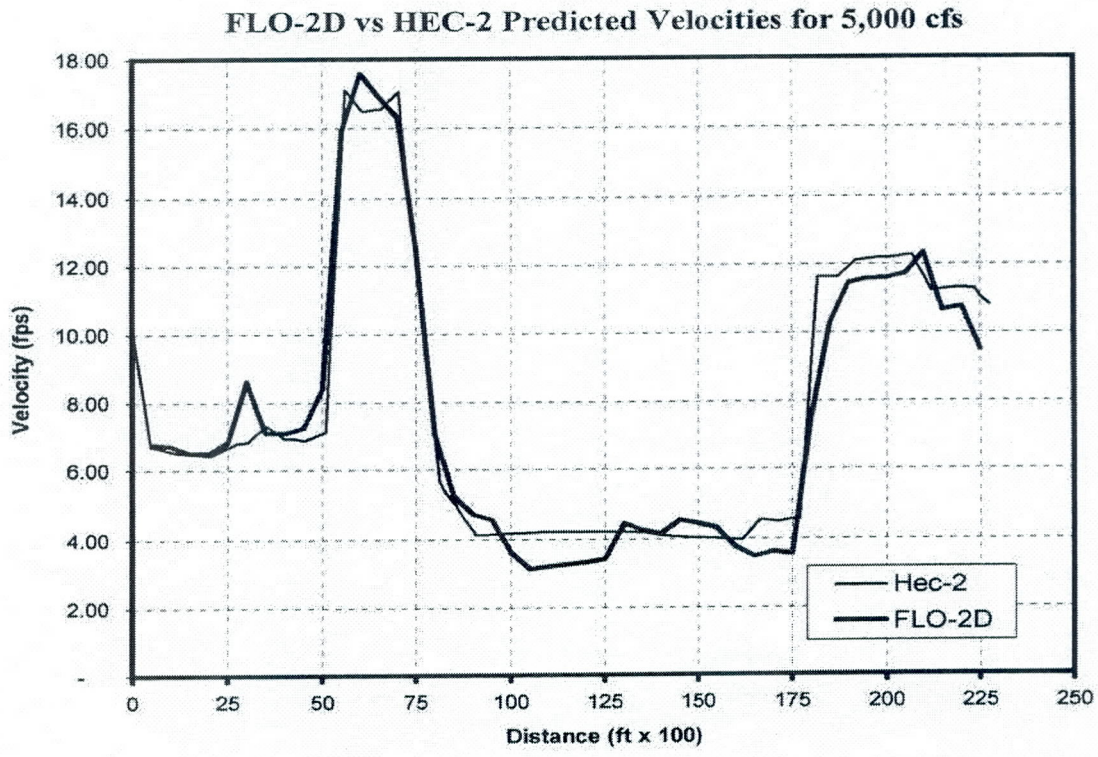


Figure 18. Comparison of FLO-2D and HEC-2 Flow Velocities for 5,000 cfs Discharge

6.4. Verification Test 3. River Floodplain Simulation - Truckee River, Reno, Nevada

The Truckee River has experienced significant flooding throughout its recorded history. In early January 1997, rainfall on snowpack and unusually warm weather conditions resulted in flooding of the Truckee River through downtown Reno, Nevada and the nearby City of Sparks. Fifteen miles of river were simulated with FLO-2D to predict the area of inundation. A extensive hydrographic data base was compiled including over 100 channel cross section surveys. Additional data for model calibration included two USGS river gages flood stage and discharge, estimated Truckee inflow hydrograph and numerous surveyed highwater marks through the inundated area. The inflow from three tributaries was estimated by the Corps of Engineers. In the Sparks Truckee Meadows area, significant overbank flooding occurred in a sparsely populated area before the river entered a bedrock canyon to the east.

The Truckee River had been repeatedly analyzed with HEC-2 but the nature of the overbank flooding in Truckee Meadows is complex. The overbank flows from the river and two tributaries commingled and the floodplain discharge was equivalent to the channel conveyance. Surging up and down the tributaries during the flood event was observed. This project provided an opportunity to review the accuracy of the FLO-2D channel-floodplain interface, the timing of overbank floodwave progression and the predicted area of inundation.

The Truckee River project task was to replicate the 1997 flood event. A total of 5,089 500 ft grid elements with 221 channel elements constituted the FLO-2D grid system. Rainfall, infiltration, 18 bridges and a number of streets and buildings were modeled to add detail to the simulation. Spatial variation in floodplain n-values and storage loss coefficients were imposed. The filling of large gravel pit was also simulated. Channel roughness values were extracted from previous HEC-2 models and calibrated to match the recorded stage at three USGS gages.

Initially the FLO-2D model results were compared with those from HEC-2 for an in-channel flow of 10,000 cfs. Overbank flow ensues at approximately 11,000 in Truckee Meadows. The predicted water surface for 10,000 cfs and the modeled bed profiles for FLO-2D and HEC-2 are shown in Figure 13. In the upstream reach, the predicted water surfaces deviate slightly where the HEC-2 cross sections are separated by a half mile or more. Other small variations in the water surface profiles occur around hydraulic structures such as weirs and bridges. Overall, the match of the water surface profiles from the two models for 10,000 cfs is excellent, especially in the sensitive flat slope at the downstream end of the reach.

The predicted area of inundation matches the flooded developed from aerial photographs and observations very well as shown in Figure 14. The gravel pit is noted by the maroon color identifying about 25 ft or more of water storage. The observed area of inundation is displayed as a black outline. A comparison of fifty surveyed high water marks and the FLO-2D predicted maximum water surface elevations resulted in an average difference of about 0.2 ft. It should be noted that both the surveyed high water marks and the estimated area of inundation were subject to the observer's interpretation.

As an indication the accuracy of the channel-floodplain interface, Figures 15 and 16 display the predicted versus measured stage reading for the two USGS gages in operation during the 1997 flood. These stage recordings represent a accurate assessment of the rise and fall of the hydrograph. The Reno gage is located about halfway through the simulated reach and the new Vista gage is located at the downstream end of the system. Upstream of the Reno gage, there is some overbank flooding and return flow from downtown Reno. The new Vista gage includes almost all of the return flow from overbank discharge in the Truckee Meadows area (the large flooded area in Figure 14). There is some ungaged inflow that reaches the new Vista gage that is not simulated in the model. Furthermore, local rainfall intensity may have exceeded the rainfall simulated in the model. Nevertheless, the FLO-2D predicted stages correlate well with the measured stage at both gages in both magnitude and timing. This is important because the progression of the overland floodwave across Truckee Meadows from both the Truckee River and Steamboat/Boynton Sloughs has a significant flood crest lag time of 6 hours or more when compared to the floodwave movement in the river. The FLO-2D model must accurately predict the volume of overbank flow, the channel flood routing, the overland flood routing and channel-floodplain exchange in order to match the stage at both USGS gages.

This project indicates that FLO-2D can be applied to river flood simulations with confidence. It demonstrates the importance of accurately assessing and correctly analyzing the relationship between channel conveyance volume and floodplain volume.

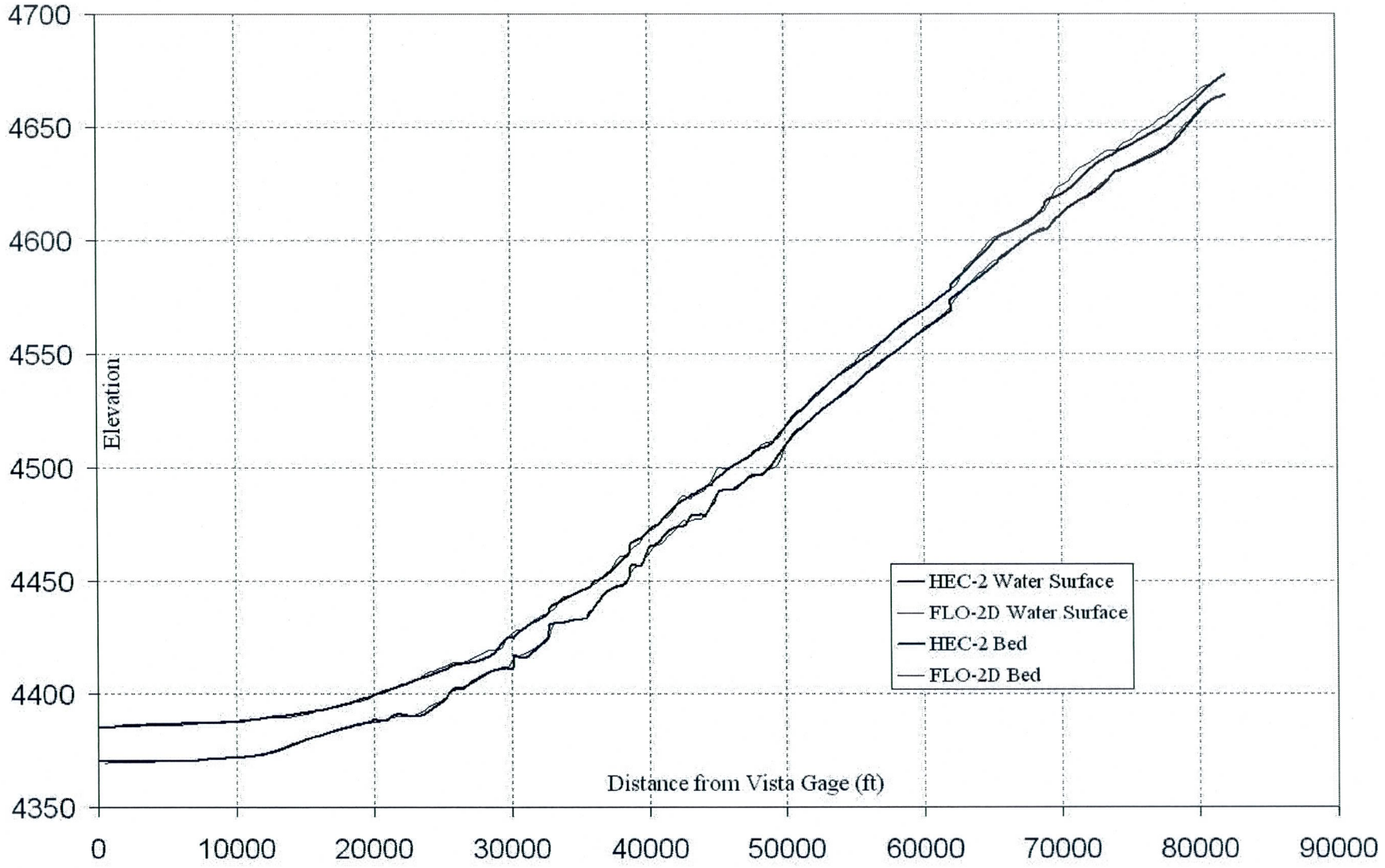


Figure 19. Predicted Truckee River Water Surface Profiles for HEC-2 and FLO-2D

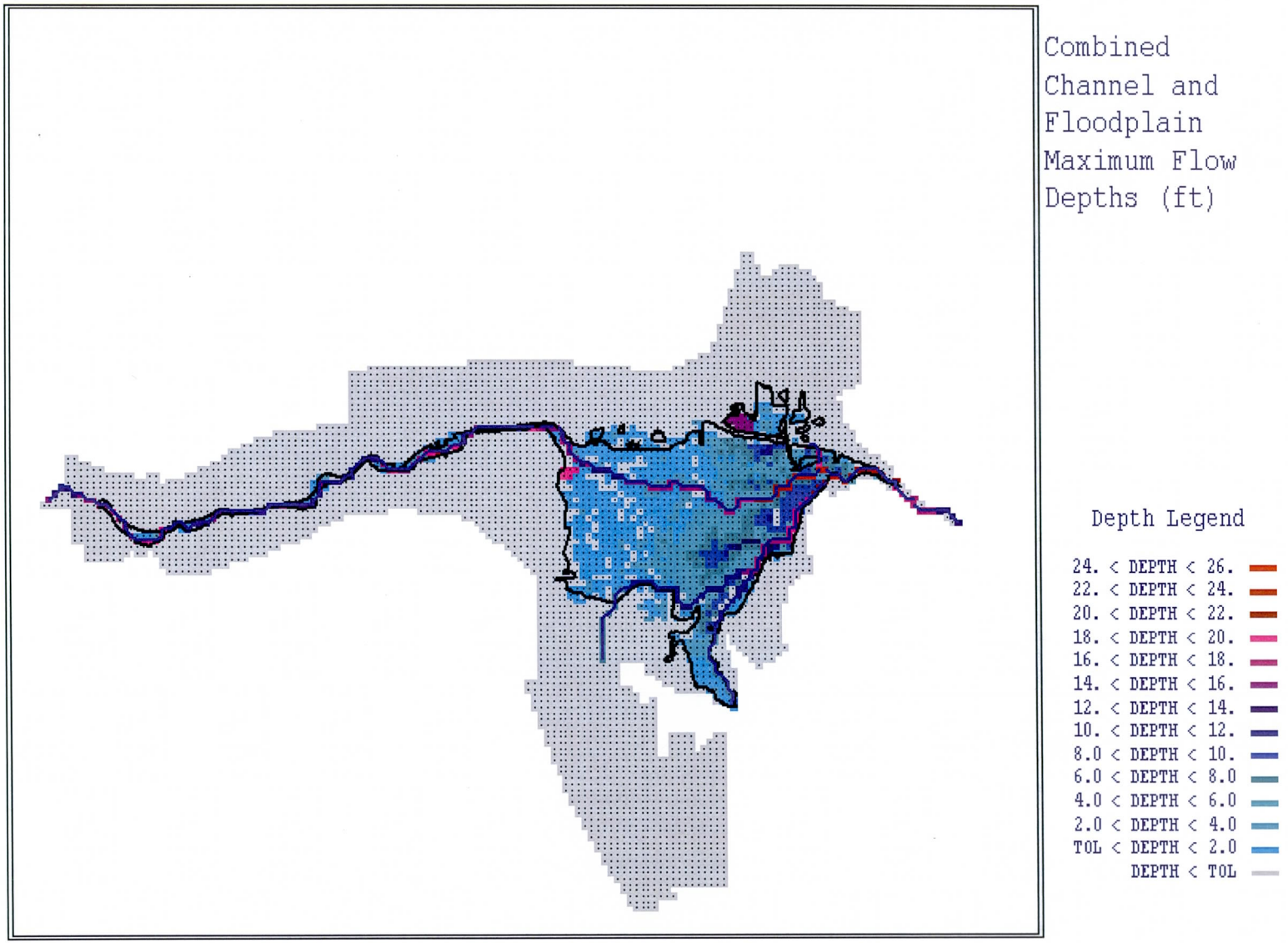


Figure 20. Truckee River Observed (Black Outline) and FLO-2D Predicted Area of Flood Inundation

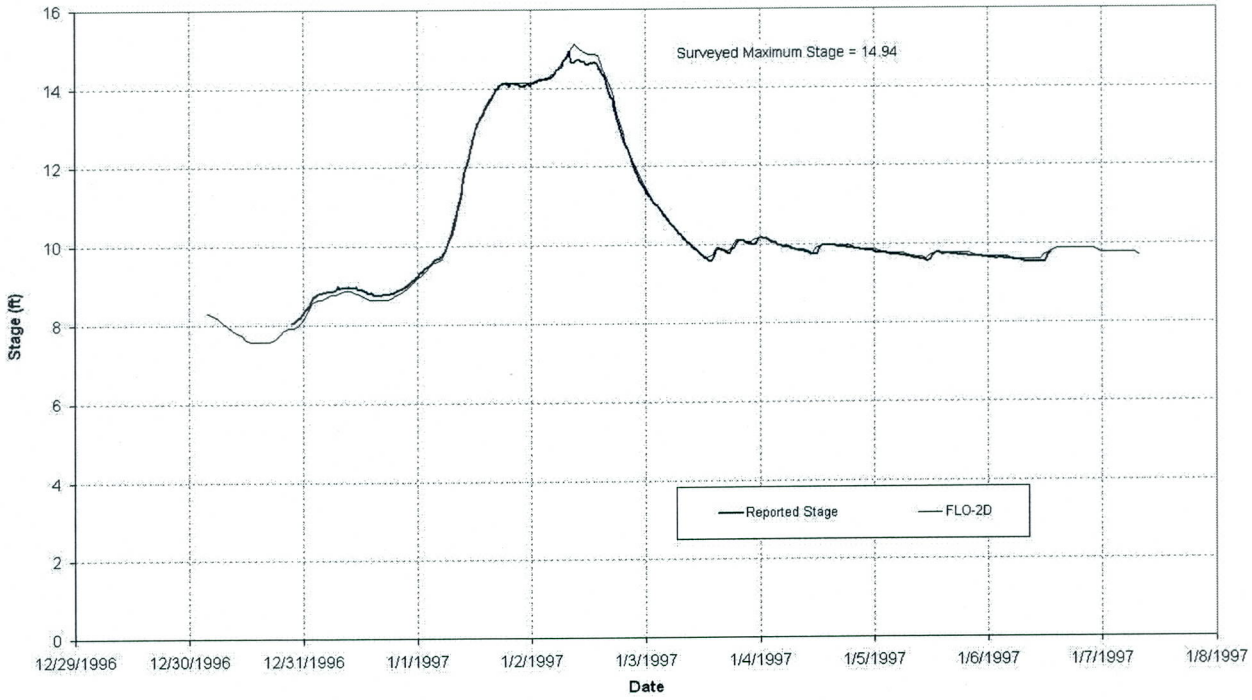


Figure 21. FLO-2D Predicted Stage for the 1997 Flood Event at the Reno Gage

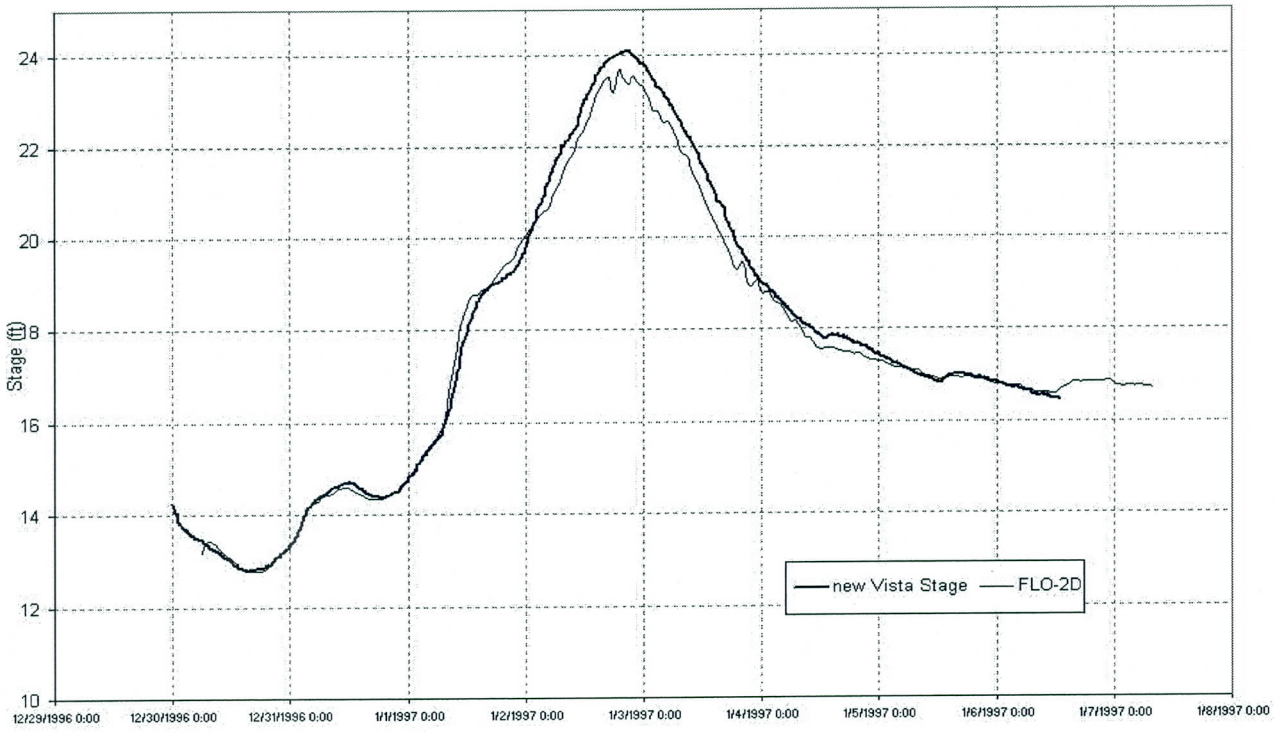


Figure 22. FLO-2D Predicted Stage for the 1997 Flood at the New Vista Gage

6.5. Verification Test 4. River Flood Routing - Green River, Utah

A discharge routing model FLO-2D was developed for the Green River system from Flaming Gorge to the Colorado River confluence, a river distance of 412 miles. The model included the six major Green River tributaries upstream to the first USGS gaging station. This project was supported by the National Park Service, Fish and Wildlife Service and the Bureau of Reclamation and it provided the opportunity to test a calibrated model with an actual discharge pulse of two days.

To develop the FLO-2D data base, forty USGS 7.5 minute maps were digitized and all the available cross section survey data was compiled from various researchers. This data was organized by river reach and analyzed for channel geometry relationships as a function of flow depth. Channel geometry was assigned to the channel elements that did not have a surveyed cross section. The inflow discharge included all the available USGS tributary inflow data, Flaming Gorge Dam releases and level logger discharge monitoring data at several locations throughout the river system.

The FLO-2D model consisted of 3,671 grid elements with 1,388 channel elements. The initial calibration run encompassed 100 days of the 1996 high flow season. Following calibration of the roughness values and infiltration parameters and the adjustment of floodplain elevations and channel geometry, the model was applied to simulate 100 days of the 1997 Green River hydrograph (Figure 17). The flood simulations included the prediction of floodplain inundation. Based on the excellent correlation between the 1997 recorded discharge data and the FLO-2D predicted discharge hydrograph at the USGS Jensen gage (95 miles downstream of Flaming Gorge) and Green River gage (293 miles downstream of Flaming Gorge), the model was applied to various flow scenarios involving regulated releases of Flaming Gorge Dam. Finally, a series of six low flow scenarios were also simulated to examine the effects of power releases on the discharge at the Jensen gage.

In November 1998, two days of power generating spike flows released from Flaming Gorge Dam (Figure 18) were used to test the FLO-2D model and predict floodwave attenuation for power operating conditions. The two spike releases of 4,400 cfs were about twice the normal daily release. Using the calibrated model, the two floodwaves were routed through entire 412 miles of river. At the USGS Jensen gage, the shape and timing of the predicted floodwaves matches almost exactly the measured hydrograph (Figure 19). The predicted flow is about 200 cfs less than the measured discharge because of ungaged tributary inflow that is unaccounted for in the model. Jones Creek in Dinosaur National Monument is responsible for most of the unmeasured flow. The Yampa River base flow contributes discharge that offsets some of the floodwave attenuation. The FLO-2D model was originally calibrated for bankfull conditions. The predicted early arrival of the two peaks at the Jensen gage can be attributed to the increased hydraulic roughness at lower flows through the steep Lodore Canyon. At low flows, numerous boulders at exposed in the canyon channel. Most of the flow spike is attenuated through the relatively flat sloped Ouray valley reach. By the confluence of the Colorado River, the two spike floodwaves had melded together in a single flood wave mound.

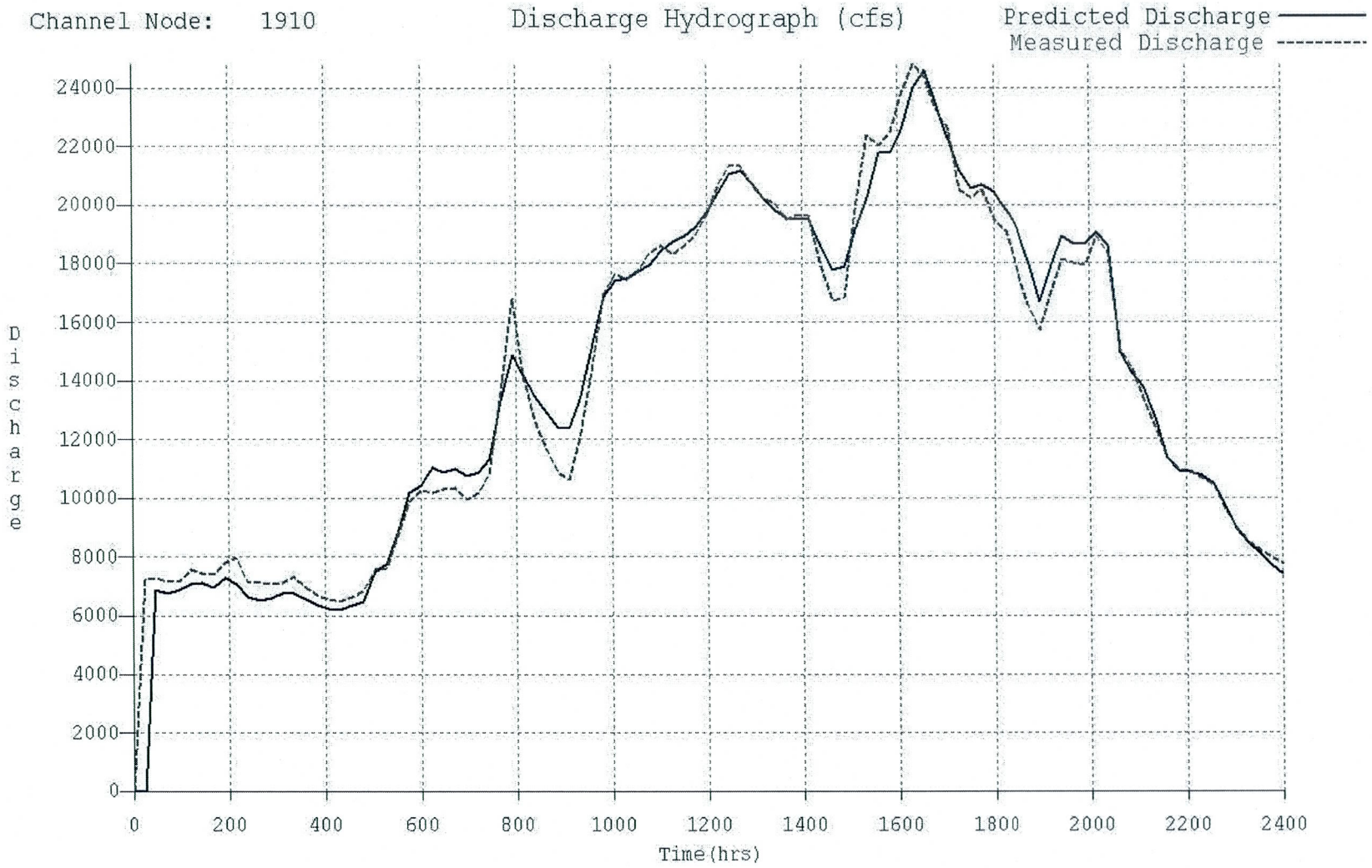


Figure 23. 1997 High Flow Season Hydrograph at Jensen, Utah (FLO-2D vs. Measured Discharge)

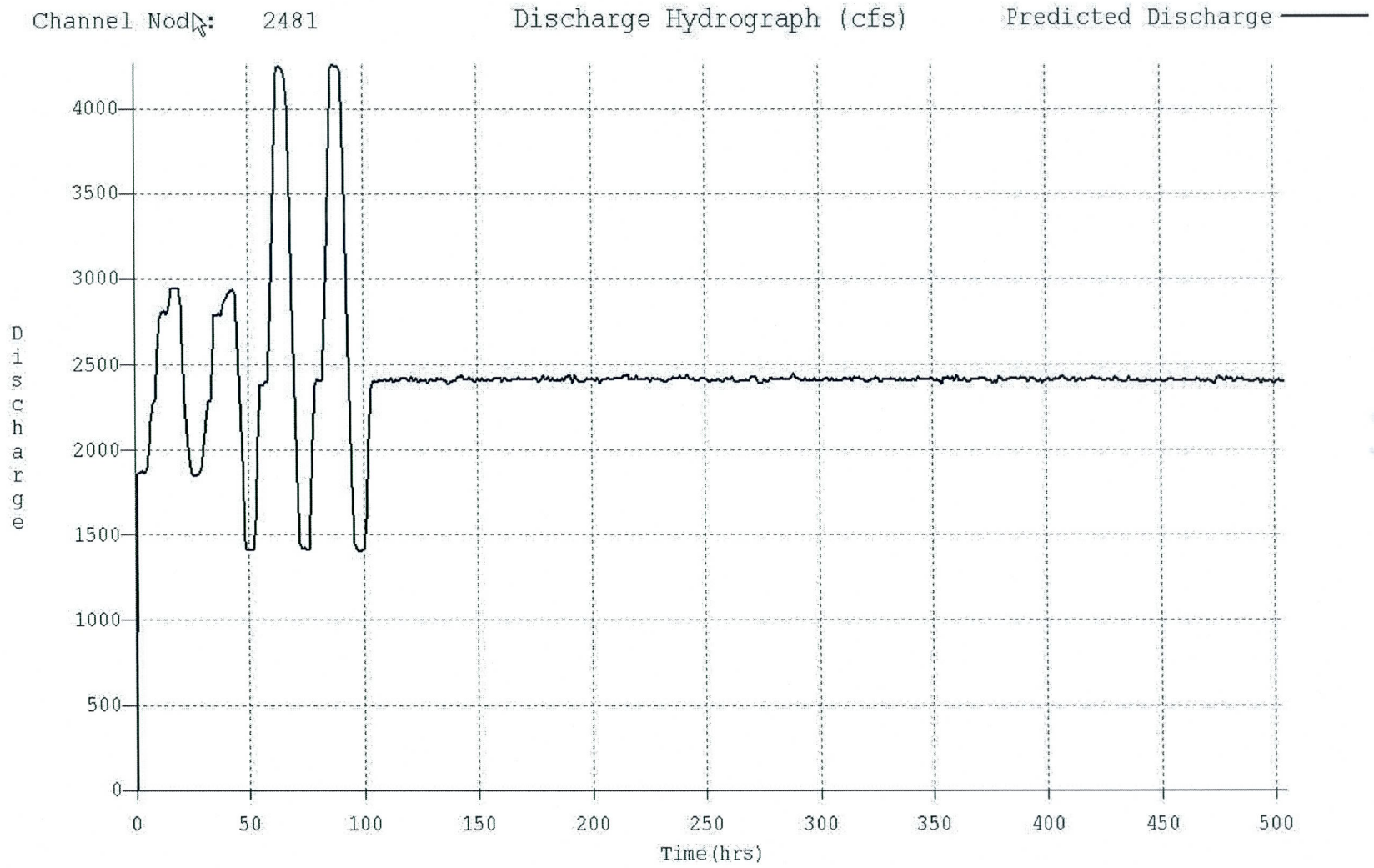


Figure 24. November 1998 Power Plant Discharge Release from Flaming Gorge Dam

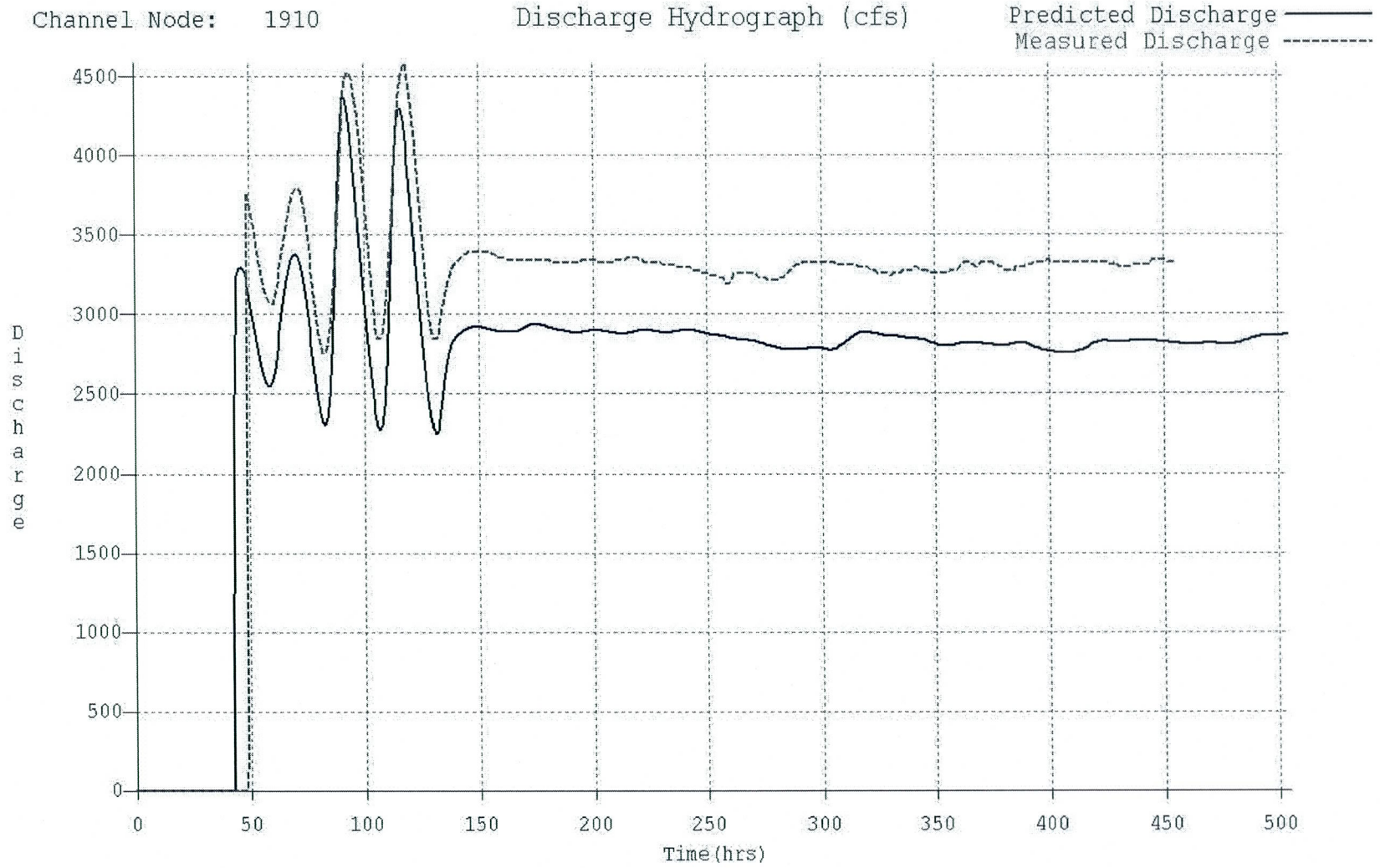


Figure 25. FLO-2D Predicted vs. Measured Discharge, Jensen USGS Gage for Power Plant Releases at Flaming Gorge Dam

By accurately predicting apriori an actual floodwave event, the FLO-2D model has been shown to be effective flood routing tool, and the channel routing algorithm has been validated. The Green River FLO-2D routing model can now be used to simulate historic or predictive flow scenarios. It can be applied to evaluate the effects of Flaming Gorge releases on flood magnitude, timing and duration to benefit fish habitat and enhance floodplain inundation.

6.6. Verification Test 5. Floodplain Inundation – Middle Rio Grande, New Mexico

The purpose of the FLO-2D model application to the Middle Rio Grande was to predict overbank flooding as function of discharge with the specific objective to quantify and locate the areas of inundation from San Acacia Diversion Dam to the San Miguel gage (approximately 48 miles). Figure 26 displays the FLO-2D predicted area of inundation of the active floodplain extending from the levee on the west to the bluff on the east. The project tasks included:

- Compilation and preparation of the cross section data.
- Creation of a digital terrain model (DTM) for the FLO-2D grid system.
- Analysis of the inflow hydrology for various historic flows.
- Calibration of a rigid bed FLO-2D model with the Bureau's 1992 inundation mapping.
- Assessment of the area of inundation as a function of discharge.
- Prediction of the area of inundation for various return period flood events.

The model calibration was based on flood inundation mapping flown by the Bureau of Reclamation on May 12, 1992. The mean daily inflow discharge at the San Acacia gage during this period ranged from 5,380 cfs to 5,720 cfs. The mean daily discharge measured at San Marcial at the end of the simulated reach ranged from 4,700 cfs to 5,230 cfs. On May 13, 1992, the mean daily discharge was 5,110 cfs at San Marcial. The FLO-2D predicted discharge ranged from 5,130 cfs to 5,220 cfs. The Bureau's mapped estimated wetted area was 2,812 channel acres and 3,014 floodplain acres at the time of the aerial photos. Comparably, the FLO-2D predicted 2,995 wetted channel acres and 3,036 floodplain acres.

This project verified that the FLO-2D model can accurately predict area of inundation as a function of discharge given an adequate channel cross section data base and an accurate river hydrograph. It confirmed that the channel-floodplain interface can accurately distribute the volume between the river and the floodplain. The actual area of inundation was mapped in detail and the river discharge was relatively uniform during the period of flooding. The FLO-2D volume conservation, overland floodwave routing, channel-floodplain interface and flow routing timing were tested in this application.

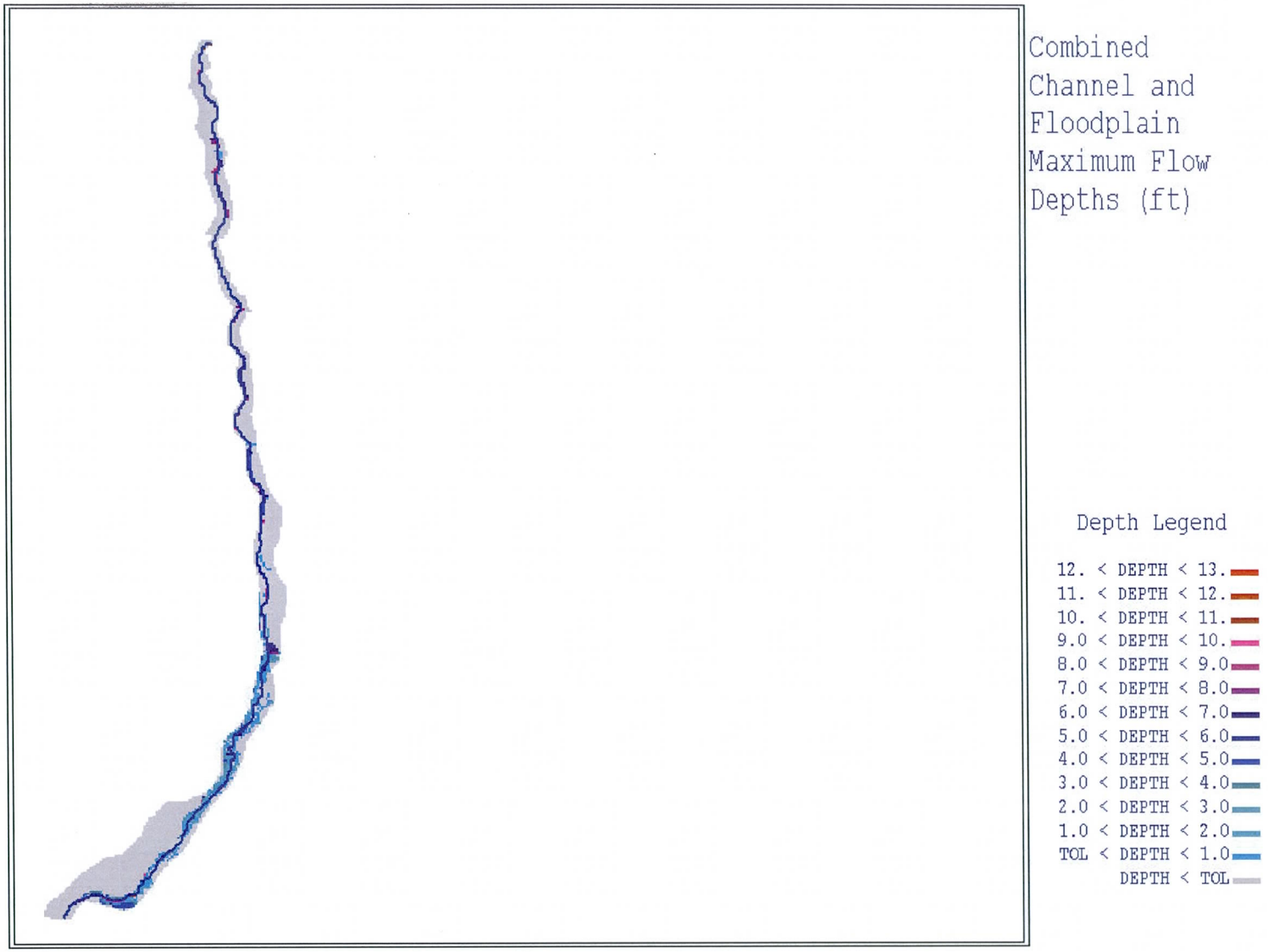


Figure 26. Area of Inundation, Middle Rio Grande, May 1992

6.7. Verification Test 6. Verification of Mudflow Hydraulics

The prediction of mudflow hydraulics requires more engineering judgment than when simulating conventional water hydraulics. The simulation of hyperconcentrated sediment flows with the FLO-2D model was verified by using field data from the 1983 Rudd Creek mudflow in Davis County, Utah (O'Brien, et al., 1993). The 1983 Rudd Creek mudflow provided the best available actual mudflow data base. The flood hydrograph and other data used in the simulation were developed by the Army Corps of Engineers (COE, 1988). The field data included:

- The area of inundation estimated from aerial photography;
- A surveyed volume of the mudflow deposit of approximately 84,000 yd^3;
- A mudflow frontal velocity on the alluvial fan of approximately the speed that a man could walk (eyewitness account);
- Observed mudflow depths that ranged from approximately 12 feet at the apex of the alluvial fan to approximately 2 or 3 feet at the debris front.

Close examination of photos taken after the event showed that the boundaries of the mudflow deposit were slightly more irregular than reported by the COE (1988) but the general area of inundation was reasonably accurate. The major portion of the flood event was over in less than seven minutes, but the mudflow continued to creep and flow over the fan for days several days.

The mudflow was initiated by a landslide and thus a relatively uniform sediment concentration was assumed. The concentration was increased slightly as the event progressed to simulate dewatering. Overland flow roughness values for the grid elements varied from 0.035 to 0.10 depending on vegetation and flow obstruction. Appropriate viscous and yield stress parameters were selected from laboratory data mudflow data. The buildings which influenced the flow path were simulated with area and width reduction factors.

A more detailed discussion of results is presented in O'Brien, et al., 1993. The maximum computed flow depth of 11.8 ft downstream of the apex compared well with the 12 ft observed depth (Figure 21). Mudflow velocities predicted on the fan ranged from 1 to 4 fps or approximately walking speed as was observed. Near the fan apex maximum predicted velocities were less than 10 fps. Just upstream of the apex in the Rudd channel area predicted velocities approached 20 fps. Predicted frontal lobe depths ranged from 2 to 4 ft depending on the location on the fan and correlated well with post-event photos. The predicted area of inundation is approximately the same as that reported by the Corps of Engineers. FLO-2D demonstrated that the streets played an important role in distributing the mudflow on the urbanized fan. A time lapse 3-D simulation of the mudflow progression over the Rudd Creek fan is presented in O'Brien, et al. (1993).

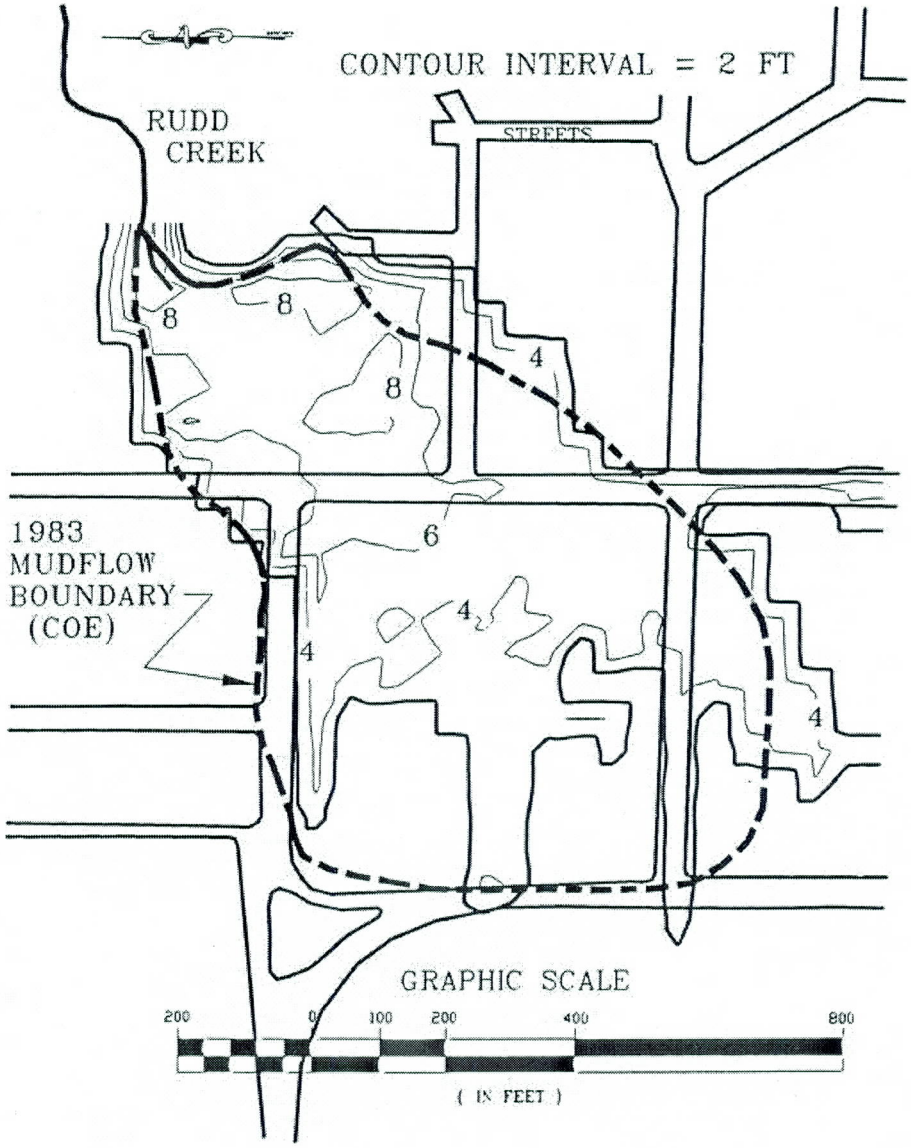


Figure 27. Rudd Creek Mudflow Maximum Flow Depth Contours

VII. FLO-2D PROJECT APPLICATIONS

7.1. Flood Hazard Delineation, Mapping and Results

FLO-2D has been applied to major river systems, very flat slope river channel and floodplains, rivers with bridges and other hydraulic structures, floodplains with levees, desert alluvial fans, mountain mudflow alluvial fans, urban floodplains and alluvial fans, large rainfall-runoff watersheds, street intersections and mudflows associated with volcanoes. It has been used for flood hazard delineation, flood mitigation design, detention basin design and FEMA FIS studies on alluvial fans. A number of projects have been conducted in foreign countries. A partial list of about over thirty FLO-2D projects is presented in the following section. This list represents only a portion of the known FLO-2D projects that have been completed.

FLO-2D results can be displayed in temporal or spatial resolution. Temporal results are available in the form of channel hydrographs, floodplain cross section hydrographs and hydraulic data (flow depths and velocities) for every grid element at specified output intervals. Channel hydrographs can be plotted for every channel element. Spatial results in the form of the maximum area of inundation, maximum flow depths and velocities can be plotted with the MAXPLOT program. In addition, there are output files that include the x- and y-coordinates along with the hydraulic variable (such as maximum flow depth or velocity) that can be imported to a CADD program for plotting results on the original mapping. With these imported files, the flow depth or water surface elevation contours can be plotted with the CADD interpretive mapping routines.

Detailed flood hazard inundation maps can be prepared by subtracting the DTM points from the grid element water surface elevations. On floodplain grid elements, the variation of the water surface elevation over a grid element is relatively minimal. By establishing the DTM points under the grid element water surface and computing the resulting flow depths, a colored contour plot of the flow depths can highlight the topographic detail within the grid element. These flow depth plots reveal flooding in ditches, depressions, over berms and in streets. All of the topographic detail that is lost in assigning the representative grid element elevation is regained through this plotting process. These color contour depth plots, when overlaid on photogrammetry maps, provide excellent detail for community flood hazard maps and FEMA FIS insurance rate maps.

7.2. Partial List of FLO-2D Projects

At the present time, FLO-2D has been applied in a number of foreign countries including: Italy, Switzerland, Austria, Mexico, Ecuador, Venezuela, South Korea, Taiwan and Thailand. A brief description of some of the FLO-2D projects completed by various agencies and consultants follows:

1. Telluride, Cornet Creek Alluvial Fan, Colorado

This first FLO-2D project was a FEMA mudflow hazard delineation study. Approximately 300 grid elements, 200 ft square were used to simulate mudflow over an urbanized alluvial fan. No buildings or streets were simulated since these components had not yet been created. The computer runs took six hours or more on a 286 MHZ computer.

2. Poudre River, Fort Collins, Colorado

After the Telluride project, a river inundation test study was attempted. Cross section data from an HEC-2 study was available for the Poudre River east of Fort Collins, Colorado. The flood inundation area was split by the I-25 embankment.

3. San Sevaine - Etiwanda Alluvial Fans, California

Alluvial fan mitigation design was simulated with the FLO-2D model. Large flood conveyance channels were designed for an urbanized alluvial fan. The model was used to establish channel geometry and freeboard for concrete lined channels. The area of inundation downstream of the channel was also predicted.

4. Hiko Springs Alluvial Fan, Laughlin, Nevada

Hiko Springs Wash is severely incised into a broad alluvial fan flood terminating at the Colorado River. Flood flows wander unconfined across the wash bottom. FLO-2D was applied to conduct a flood hazard delineation study for a FEMA CLOMR. The model was also used to predict velocities and depths along a floodwall where sediment deposition was considered to be a potential problem. The model predicted flow accumulation along the floodwall, scour not deposition turned out to be the critical concern and the wall was buried eleven feet below the ground surface with footers.

5. Whiskey Pete's Alluvial Fan, Stateline, Nevada

An alluvial fan flood hazard delineation study was conducted above a casino resort. Flows over the alluvial fan were collected at railroad berms and directed into culverts that could overtop the railroad embankment. The culvert outflows were directed at the casino. A concrete channel was designed to collect the flows upfan of the casino. Sediment transport was analyzed to determine the potential loss of channel conveyance laterally across the fan. Flow runup in the channel was a design consideration. The Desert Research Institute in Las Vegas also analyzed the Whiskey Pete's flows with FLO-2D to determine the potential for overtopping the railroad embankment. A master's thesis was written on the application of FLO-2D and other flood models.

6. Barnard Creek Mudflow Alluvial Fan, Centerville, Utah

Barnard Creek is a completely urbanized alluvial fan with a small debris basin at the fan apex. FLO-2D was applied to delineate the potential water flood and mudflow hazard on the fan. Streets, buildings and the debris flow overflow were simulated. The flood scenarios included water flooding, rainfall, mudflow, and rainfall and mudflow. The Barnard Creek data files are provided as one of the example data file sets.

7. Rudd Creek Mudflow Alluvial Fan, Centerville, Utah

The 1983 Rudd Creek mudflow was well documented. The data base was used to calibrate some of the mudflow parameters in the FLO-2D model including the laminar flow resistance. The inflow hydrograph, area of inundation, maximum flow depths and velocities and flow cessation depths were known or observed. Street flow and building obstruction were simulated.

8. Mount St. Helen, Volcano Observatory, Cascade, Washington

The Pine Creek mudflow on the south face of Mount St. Helen was simulated with FLO-2D. A comparison of volcanic mudflow parameters measured and computed by the Volcanic Observatory was made. The floodwave time to peak and predicted flow depths and velocities were compared with field observations.

9. Broadmoor Alluvial Fans, Colorado Springs, Colorado

Expensive residential units were planned for a series of steep coalescing alluvial fans along the front range of Colorado Springs that had experienced numerous mud and debris flows in the past. Large boulder levees were evidence of the severity of the potential debris flow events. Flood hazard delineation and mitigation design were based on FLO-2D mudflows simulations.

10. Pima County, Tucson, Arizona

At the request of a local consulting firm, a FLO-2D application to an alluvial fan that was planned for development demonstrated the model's capability to predict distributary flows over a broad alluvial fan surface.

11. Monroe Creek Alluvial Fan, Richfield, Utah

Monroe Creek flows over a large alluvial fan with a significant supply of boulders from the upstream watershed. The Corps of Engineers used FLO-2D to conduct an unconfined flood simulation of overbank flows. Rectangular, trapezoidal and natural shaped cross sections were used to represent the channel geometry. Overbank flooding and return flows to the channel were simulated to delineate the flood hazard. The data files for are included on the model as an example channel application.

12. Arroyo Pasajaro Alluvial Fan, Colinga, California

In March 1995 a significant flood filled the detention basin along the California Aqueduct forcing the release of water onto the alluvial fan downfan of the Aqueduct. The State of California was sued by flooded farmers who lost crops and had buildings inundated. The Department of Water Resources contracted to simulate the flooding from the detention basin outlet culverts. Storm rainfall was also modeled. The results demonstrated that the flooding was due the rainfall first, then culvert releases. Both flood sources resulted in essentially the same area of inundation.

13. Rio Grande, Cochiti Reservoir to Elephant Butte Reservoir, New Mexico

A FLO-2D model of the Middle Rio Grande was developed for 173 river miles using 29,782 500 ft grid elements. In addition to flood inundation flows, an investigation was conducted to determine flooding potential for Bosque restoration flows. Floodwave attenuation was an important facet of this study. Levees were simulated. The Bureau of Reclamation, Corps of Engineers and the Fish and Wildlife Service participated in the study

14. Green River Routing, 412 Miles Flaming Gorge to Colorado River, Utah

This Green River flood routing study was a cooperative project between the Bureau of Reclamation, Fish and Wildlife Service and National Park Service to develop a river discharge routing model and assess overbank flooding for endangered fish nursery habitat. The FLO-2D grid system was later extended up the Green River tributaries to the first USGS gaging stations. An excellent match was obtained between the predicted hydrograph and the USGS recorded discharges in the lower Green River.

15. Vicee Canyon Alluvial Fan, Carson City, Nevada

Vicee Canyon watershed debouches into an incised wash on the alluvial fan. A small debris basin was constructed at the terminus of the alluvial fan. FLO-2D simulated flows in the wash and through the debris basin. Both the existing basin and the proposed expanded basin were found to be inadequate for the 100-year event.

16. Glenwood Springs and Ouray Mudflow Alluvial Fans, Colorado

The Cities of Glenwood Springs and Ouray are situated on a series of coalescing alluvial fans. The Sacramento District of the Corps of Engineers applied FLO-2D to delineate the water flood and mudflow hazard on these two urbanized alluvial fan areas for a FEMA flood insurance study. For the Glenwood Springs project, about 50 small watersheds and their alluvial fans with the city limits were simulated. In Ouray, two existing concrete channels were modeled.

17. Wailupe Creek, Hawaii; Corps of Engineers

The Pacific District of the Corps of Engineers conducted an urban alluvial fan flood hazard delineation study. Several channels flowed through the urban area that collected flows and debouched into the ocean. Streets and buildings were modeled to assess the area of inundation.

18. Aspen Mountain, Aspen, Colorado

WRC Engineering, Inc. of Denver, Colorado performed an alluvial fan flood and mudflow hazard delineation study for the City of Aspen. Several watersheds contributed to the flow the city. Mapping involved the assessment of street flow and building flow obstruction. Tetra Tech later expanded this study for development mitigation.

19. Diamond Alluvial Fan, Las Vegas

PBSJ of Las Vegas conducted an alluvial fan rainfall/runoff flood study above a proposed development. Rainfall was simulated in the upper basin and runoff was routed to the development site. This project represented a good example of simulating the entire hydrology of the basin and fan complex.

20. Lehi and Nephi Alluvial Fans, Utah

The Corps of Engineers, Sacramento District conducted two alluvial fan flood hazard delineation studies involving culvert flows under an interstate highway. The culvert capacity was an integral part of the flood study involving culvert plugging, ponding and discharge onto an unconfined surface.

21. Truckee River, Reno, Nevada

In January 1997, the Truckee River flooded downtown Reno and the surrounding area. The Sacramento District of the Corps of Engineers conducted a river flood hazard delineation study involving 15 miles of the river and all of the urbanized floodplain. Streets, buildings and the flow through 18 bridges were modeled. The analysis included replication of the 1997 flood including predicting river stages and discharge, area of inundation and water surface elevations on the floodplain. A full compliment of flood frequency events and project flood mitigation conditions were modeled.

22. Section 30, Paradise Alluvial Fan, Las Nevada

DeRoulhac Consulting, Inc. conducted a detailed unconfined flood simulation for an alluvial fan upfan of a proposed subdivision.

23. McCoy Wash, Blythe California

The NRCS applied FLO-2D downstream of a proposed detention basin spillway to assess the project design flooding over an unconfined surface with the flow eventually returning to McCoy Wash. The FLO-2D sediment transport component was applied to determine the scour and deposition on the alluvial fan surface before the flow re-entered the wash.

24. Roaring Fork Club Alluvial Fans, Basalt, Colorado

A golf course was constructed on a series of coalescing alluvial fans on both sides of the Roaring Fork River. A channel mudflow hazard delineation was conducted and mitigation measures were recommended. Several cabin sites including one under construction were moved on the basis of the FLO-2D simulations.

25. Reata Pass, San Bernardino, California

Tetra Tech, ISG analyzed flows on an alluvial fan in conjunction with the design of a concrete conveyance channel. Overflows from the channel were released into an alluvial fan channel for the purpose of maintaining the desert ecosystem on the alluvial fan.

26. Frazier Creek Alluvial Fan, Strathmore, California

Flooding of Frazier Creek inundated several property owners who claimed that the flooding was the result of upfan berms and channel improvements constructed by ranchers. FLO-2D was used to simulate historic conditions of the fan as they existed in 1951 before the improvements were made. Then the model was applied to the existing conditions in support of a lawsuit filed by the flooded property owners.

27. Howes Street Intersection, Fort Collins, Colorado

Lidstone and Anderson Consultants of Fort Collins used small grid elements of ten feet to simulate flows within a large street intersection in Fort Collins. The purpose of applying the FLO-2D model was to predict the discharge split between the various streets at the intersection to design storm runoff.

28. Guagua Pichincha Volcano Mudflows, Quito, Ecuador

Mudflows through the capital city Quito were simulated to prepare a risk assessment plan for Quito's government and the United Nations Development Program. Streets and buildings were modeled in flood scenarios involving volcanic ash mudflows of varying concentrations.

29. Avila Mountain Mudflows, North Coast, Venezuela

In December 1999, the watersheds along the North Coast of Venezuela flooded under intense prolonged rainfall approaching a 250 year return storm. Mudflows and flooding resulted in over 12,000 deaths. FLO-2D was used to simulate the 1999 floods and mudflows on a number of alluvial fans. It was then applied to test potential mitigation measures.

30. Orestimba Creek, near Neuman, California

The 1995 flood of Orestimba Creek in the north Central Valley was simulated to conduct a flood delineation study. Railroad embankment and county roads were simulated as small levees. Orestimba Creek entered the San Joaquin River with a mild slope. This project was conducted for the Sacramento District Corps of Engineers.

31. Red Arroyo near Alamogordo, New Mexico

The Albuquerque District Corps of Engineers conducted an alluvial fan and arroyo flood investigation to determine the mitigation effects of a proposed flood retention basin. The flooding included distributary channels.

32. Skunk Creek Wash, Maricopa County, Arizona

A pair of alluvial washes were controlled by a retention facility with an overflow spillway. A FLO-2D model was developed by Tetra Tech ISG to simulate the flood distribution within the retention pond. The berm and spillway were simulated. The ponded flood waters overtopped the berms in several places for the design flood event. Mitigation measures were proposed.

33. Lower Mission Creek, Los Angeles, California

Lower Mission Creek winds through an urban area before entering the ocean. A private development was elevated and was apparently out of the flood hazard. HEC-2 and HECRAS were ineffective in the urban area. FLO-2D was applied by a consultant with assistance from Tetra Tech ISG to examine the flood hazard in detail by simulating the highway and railway embankments and the loss of storage due to the numerous buildings.

34. Glenwood Springs Hospital Expansion, Glenwood Springs, Colorado

The Glenwood Springs Hospital is located on a series of coalescing alluvial fans created by mudflows. Mudflow mitigation for the hospital expansion was analyzed with the FLO-2D model. Mudflow through an urban area with streets on a steep alluvial fan was simulated.

35. Whitewater Alluvial Fan, near Blythe, California

Tetra Tech ISG conducted an extensive alluvial fan study with the sediment component to determine the effects of a berm to redirect flood flows in a National Wildlife Refuge.

36. West Stanislaus Project, Central Valley, California

A series of interlacing river channels across the Central Valley were analyzed by the Corps of Engineers. FLO-2D was applied to assess overbank flooding that entered other river channel segments as return flows.

37. Rogue River, Oregon

Lidstone and Associates of Fort Collins conducted a complex multiple channel flood inundation study of the Rogue River that included gravel pits on the floodplain. Both split flow and channel confluence flows were simulated.

VIII. REFERENCES

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

Akan, A.O. and B.C. Yen, 1981. "Diffusive-wave flood routing in channel networks," J. of Hyd., ASCE, 107(6), 719-732.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

MacArthur, R.C. and D.R. Schamber, 1986. "Numerical methods for simulating mudflows," Proc. of the 3rd Intl. Symp. on River Sedimentation, Univ. of Mississippi, 1615-1623.

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

National Research Council, 1982. "Selecting a methodology for delineating mudslide hazard areas for the National Flood Insurance Board on the Built Environment," National Academy Press, Washington, D.C.

National Research Council, 1983. "An evaluation of flood-level prediction using computer-based models of alluvial rivers," National Academy Press, Washington, D.C.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

INPUT DATA
FORMAT AND DESCRIPTION

FLO-2D INPUT DATA OVERVIEW

The FLO-2D data input consists of a series of ASCII files organized by model components. A flood model begins with routing a hydrograph over unconfined topography and then can be expanded with channel flow or street flow. As the model builds, the flood simulation can be enhanced by adding rainfall, infiltration, hydraulic structures, levees, mudflows, sediment transport, rills and gullies, buildings and flow obstructions. The FLO-2D options are initiated through on-off switches found in the control data file (CONT.DAT). If the options are turned on, then the appropriate data files must be created. For example, if rainfall is selected as a simulation option then a RAIN.DAT file must be prepared. This manual describes the various data file variables and the file format.

To conduct a FLO-2D flood simulation, the finite difference grid system must be created first. The data files FPLAIN.DAT and CADPTS.DAT contain the topographic and grid system data and can be automatically created (error free) using the grid developer system (GDS) program. There are six required files to conduct a basic overland flow simulation: CONT.DAT, TOLER.DAT, FPLAIN.DAT, CADPTS.DAT, INFLOW.DAT and OUTFLOW.DAT. Once the FPLAIN.DAT and CADPTS.DAT files have been created, the user can focus on the remaining three required files. CONT.DAT and TOLER.DAT contain control and numerical stability data and consist of only a few lines of data. INFLOWDAT contains the inflow hydrograph and OUTFLOW.DAT contains outflow nodes and outflow control. The user is encouraged to start with a simple overland flow simulation and build the flooding detail into the model one component at a time to observe the effects of each feature. There are several pre-processor programs such as FLOENVIR and PROFILES that can assist in developing some of the data files. These processor programs are described in the following sections.

From the graphical user interface (GUI – pronounced gooey), the user can execute the FLO-2D model and processor programs from a pull-down menu. A FLO-2D flood simulation can also be initiated by copying the FLO.EXE model program into a project subdirectory and double-clicking on the file name in Explorer. When the model is running the user has the option watching the flood progression over the grid system in a graphics routine. This graphics routine also displays the inflow hydrograph and the rainfall on the grid system. Upon completion of the flood simulation, there are post-processor programs (MAPPER, MAXPLOT, PROFILES and HYDROG) that will display some of the results. FLO-2D can generate huge output files so the user is cautioned to use discretion in deciding what output data may be valuable.

This Data Input Manual is 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 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 User's Manual.

TABLE of CONTENTS

Page

FLO-2D INPUT DATA OVERVIEW i

I. FLO-2D INSTALLATION AND GETTING STARTED 1

1.1 General 1
1.2 FLO-2d Installation 1
1.3 Uninstalling the FLO-2d Software 2
1.4 Start a New Project 2
1.5 Begin the Model Simulation 2
1.6 Save the Work as You Go 2
1.7 Some Keys to Getting Started 3
1.8 Things to Watch For 3

II. CREATING AND EDITING DATA WITH THE GUI 6

2.1 Open the GUI 6
2.2 Opening Data Files or the 'difference between round and rectangular buttons' 6
2.3 Editing the GUI Data Files 6
2.4 Running FLO-2D from Explorer 8
2.5 Starting from Scratch on a New Project 8
2.6 Metric Option 11
2.7 FLO-2D Example Flood Simulations 11
2.8 Exporting Topographic Data from CADD Programs 13

III. PRE-PROCESSOR PROGRAMS 14

3.1 Introduction 14
3.2 FLOENVIR 14
3.3 GDS 18

3.3.1 GDS System Overview 18
3.3.2 Getting Started on the Grid Developer System 20
3.3.3 Installation of other Required DLL Files 20
3.3.4 Data Requirements 20
3.3.5 Developing the Grid System 21
3.3.6 Create a Grid System by Digitizing Points 23
3.3.7 Create a Grid System from DTM Points 25
3.3.8 Create a Grid System from ArcInfo ASCII Grid File 25
3.3.9 Create a Grid System from an Existing FLO-2D Project 26
3.3.10 Grid System Generation 26

TABLE of CONTENTS (cont.)

Page

3.3.11 Assigning Manning's Roughness Coefficients (n-values) 30
3.3.12 Interpolation of Terrain Elevations 31
3.3.13 Detailed Description of GDS Menus and Commands 34

3.4 Profiles 89

IV. INPUT FILE DESCRIPTIONS 95

4.1 General 95
4.2 List of Programs Files Units 96
4.3 Data Files 97

Control Data Files

CONT.DAT 98
TOLER.DAT 104

Floodplain Data Files

FPLAIN.DAT 108
INFLOW.DAT 111
OUTFLOW.DAT 114
RAIN.DAT 120
INFIL.DAT 123
EVAPOR.DAT 126

Channel Data Files

CHAN.DAT 128
XSEC.DAT 139

Miscellaneous Data Files

HYSTRUC.DAT 141
STREET.DAT 146
ARF.DAT 151
MULT.DAT 154
SED.DAT 157
LEVEE.DAT 160

TABLE of CONTENTS (cont.)

Page

V. DATA ARRAY SIZE LIMITATIONS AND VARIABLE RANGE....165

VI. OUTPUT FILES AND OPTIONS....171

VII. POST-PROCESSOR PROGRAMS....180

7.1 HYDROG....180
7.2 MAPPER....183

7.2.1 Introduction....183
7.2.2 Installation and Starting the Mapper Program....184
7.2.3 Installation of Other Required DLL Files....185
7.2.4 MAPPER Menu and Toolbar Commands....185
7.2.5 Plot Examples....209

7.3 MAXPLOT....213
7.4 PROFILES....216

VIII. CREATING FLOW DEPTH CONTOURS WITH CADD PROGRAMS....220

IX. DEBUGGING AND TROUBLESHOOTING THE DATA FILES....222

APPENDIX. OBSOLETE PRE-PROCESSOR PROGRAMS....229

I. FLO-2D INSTALLATION AND GETTING STARTED

1.1 General

FLO-2D was written in Lahey Fortran 95®. Depending on your computer speed and the project application, the flood simulation might have a runtime ranging from 5 minutes to 24 hours. To apply FLO-2D, a Pentium chip computer with at least 16 Megs of RAM is recommended and a faster/bigger computer is better. Approximately 100 Megs of disk space will accommodate the model, the graphical user interface (GUI), processor programs, input data and output files for a project. FLO-2D can be run from the GUI in the Windows 98/2000/NT®. It can also be initiated in the DOS operating system, if necessary.

There are several processor programs that can be used to graphically edit or create spatially variability in the data base. The GUI was developed to simplify the ASCII text data input. The GUI is easy to use and a few instructions are listed in the following section to assist in the GUI installation and application. After the basic data files have been created, the grid system attributes can be edited in the FLOENVIR or the GDS processor programs. The FLOENVIR allows you to select individual grid elements or groups of elements to spatially edit the component data such as infiltration, streets, levees, etc.

1.2 FLO-2D Installation

FLO-2D is compatible with the MS-Windows™ operating systems including versions Windows 95/98™, Windows NT™, Windows 2000™ or Windows XP™. Recommended minimum computer requirements are 128 MB RAM and 500 MB of available hard disk space. A high resolution color monitor, a mouse and a graphical output device are necessary. Most of the FLO-2D and processor program numerical computations are only limited by the available RAM memory. Projects with a large DTM point data base or a selected large grid system can put a high demand on the computer resources and computations may take several minutes. Faster is better and a computer with a PENTIUM IV 2.0 GHz or faster processor is suggested for the GDS program.

To load the FLO-2D package onto the computer hard drive, put in the CD and wait for the installation process to begin. If after a few seconds, the installation process did not automatically start, then click on the START button at the bottom of your screen and click on the RUN and BROWSE buttons to locate the SETUP.EXE file on the CD. Follow the installation instructions as they appear in the dialog boxes on your screen. You will need to select a subdirectory for the installation. The default is C:\FLO-2D. The FLO-2D model, the manuals and all the processor programs are loaded into this main subdirectory. Several additional subdirectories are also created; one subdirectory with the help text files (flo_help) and several subdirectories with example project files (e.g. BARN, MONROE, DIAMOND and TRUCKEE).

1.3 Uninstalling the FLO-2D Software

You can remove the FLO-2D program and all of it's attendant software from your computer with the Windows standard *Remove Program* procedure. When removing the model, it is extremely important to choose the ***keep shared DLL/OCX*** option and ***not to remove them from your system***. GDS uses several shared components that may be essential for the Windows operating system and for other applications.

1.4 Start a New Project

Start by creating a subdirectory for the project data files and import the DTM data base files, map images and aerial photos to the project subdirectory. Open the GDS from the GUI or by double clicking on the GDS.EXE file in the FLO-2D subdirectory. Follow the GDS instructions presented later in the manual to generate the FPLAIN.DAT and CADPTS.DAT files. After the FPLAIN.DAT and CADPTS.DAT files have been developed, create the CONT.DAT and TOLER.DAT files. You can create the CONT.DAT and TOLER.DAT files from scratch in the GUI or an ASCII editor such as WordPAD to develop the files using format in data file descriptions later in the manual. The INFLOW.DAT and OUTFLOW.DAT files must also be created to start the flood simulation.

1.5 Begin the Model Simulation

After the six required data files have been created, FPLAIN.DAT, CADPTS.DAT, CONT.DAT, TOLER.DAT, INFLOW.DAT and OUTFLOW.DAT, a basic overland flood simulation can be conducted. Double click on the FLO-2D icon to initiate the GUI. Click on the CONT.DAT large button, locate the project subdirectory in the window and click on the Open button. You must have at least one data file open (e.g. CONT.DAT file) to run the model. Edit the files as necessary, save the file changes and then click on the EXECUTE button and the FLO.EXE name in the top menu to start the model simulation. Alternative method for initiating the model is to copy the FLO.EXE file into the project subdirectory and in Explorer double clicking on the FLO.EXE file name to start the flood simulation.

1.6 Save the Work as You Go

Until you are familiar with the GUI, FLOENVIR, PROFILES or FLO-2D data files it is suggested that you frequently save the data files as they are being developed to avoid losing any data input due to an errant click of a mouse button.

1.7 Some Keys to Getting Started

Grid System

- ✓ If you add grid elements to FPLAIN.DAT, the CADPTS.DAT file will have to be edited.
- ✓ The grid system must begin with grid element #1 and be consecutive with no missing grid element #'s.
- ✓ There should be no dangling grid elements connected only by a diagonal.

Channel Flow

- ✓ Use the full dynamic wave momentum equation (KMODEL = 3) for channel flow.
- ✓ The CHAN.DAT file channel elements should be organized from upstream to downstream and the elements must be continous.
- ✓ At a channel confluence, the main channel grid element in the downstream direction must be lower in elevation than the confluence element.
- ✓ Eliminate channel elements with a channel length (XLEN) less than 50% of the grid element side width. Connect these channel elements by the diagonal.
- ✓ Create a positive bed slope at channel inflow and outflow nodes.

Inflow/Outflow Nodes

- ✓ Inflow and outflow nodes should not have other components such as hydraulic structures, streets, ARFs, etc.

1.8 Things to Watch For

The following is a list of key model issues that may arise and what to do about them.

Channel Surging

Channel surging may numerical or represent physical response to hydraulic conditions. It will appear as spikes in the discharge hydrographs of a few elements and may be viewed in the HYDROG program hydrograph plots or in the CHANMAX.OUT file. Generally, adjusting the relationship between the channel flow area, n-values and slope between these channel elements to better reflect reality will eliminate the surging. Avoid abrupt transitions in flow areas between contiguous channel elements.

Sticky Grid Elements

A review of the TIME.OUT file will reveal which grid elements are causing the model to slow down. In the case where one or two grid elements are causing most of the timestep decrements, some modification of the elevations and n-values around these elements will speed up the model. This is also true for sticky street elements.

Limiting Froude Numbers

When the limiting Froude numbers for floodplain, channel or street grid elements is exceeded, the roughness n-values is incremented by 0.001 and the change is reported in the ROUGH.OUT file. If the maximum n-values in this file appear to be unreasonable, the flow area, slope or n-values should be revised.

Channel Slope Adjustments

Adverse channel slopes can be simulated by FLO-2D. Smoothing out an irregular slope condition over several channel elements to represent average slope conditions will speed up the simulation. Surveyed cross sections with scour holes can result in local adverse slopes than can misrepresent the average hydraulic conditions that the model is simulating.

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.

Floodplain Surface Area Reduction

The distribution of flood storage on the grid system can be influenced by assigning area reduction factors (ARFs). For large flood events, the global assignment of ARF values is usually sufficient. For flooding detail in a local project area, individual grid element ARF assignment may be justified.

Edit Topography

CADD programs can interpolate and assign elevations to grid elements that do not represent the average floodplain surface. A highway on-ramp with a grid element is a classic example where the on-ramp elevation may not be indicative of the potential flood storage on the grid element. The grid developer system (GDS) was designed with filters to eliminate this problem. Use the GDS and FLOENVIR to review and edit floodplain grid element elevations.

Spatial Variation of n-values

Generally the spatially variation of n-values will affect the floodwave progression but will not significantly impact the area of inundation if the flood has a long duration and covers a large area. Concentrate on the project area when adjusting n-values, but use the AMANN variable in CONT.DAT to make global n-value modifications.

Street Flow

In urban areas with shallow flow, streets are important conveyance features for distributing the flow over the grid system. Street detail is important to the flood hazard delineation in urban areas.

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

Hyperconcentrated Sediment Flows (Mud and Debris Flows)

For desert alluvial fans with a sandy surface, 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. Do not invoke the mudflow component (MUD = 1 in CONT.DAT) unless sediment concentrations greater than 20% by volume are expected. Use the XCONC factor in the CONT.DAT file to bulk desert alluvial fan flows by 10% to 15 % by volume. Do not set both the MUD and ISED switches to "on" in the CONT.DAT file in the same simulation.

Model Calibration and Replication of Flood Events

Estimated flood hydrology (both rainfall and flood hydrographs) can represent a significant departure from reality when replicating historical floods. When attempting to match measured flood stages, high water marks or channel discharges, focus first on obtaining a reasonable estimate of the flood volume, then concentrate on the model details such as n-values, ARFs and street flow. Flood volume is more important to flood routing than peak discharge.

II. CREATING AND EDITING THE DATA FILES WITH THE GUI

2.1 Open the GUI

Click on the FLO-2D icon to open the GUI, and then click on the large, rectangular button with the CONT.DAT name to access a subdirectory listing. Select the CONT.DAT in the project subdirectory and click OK. You can also click on the file menu to access a pull down menu for opening and closing files. Alternatively, you can open the GUI by double clicking on FLO-2D.EXE in the FLO-2D subdirectory. By clicking on the river image as the GUI is loading, you can close the image and speed up the GUI loading.

2.2 Opening Data Files or the 'difference between the round and rectangular buttons'

There are 15 sets of data files listed on the left side of the main window. Each data file is represented by two buttons: a small round (radio) button and a rectangular button.

To open existing data files, use the rectangular button with the data file name:

Clicking on the rectangular buttons will access the directory for selecting existing files. Select the correct directory and file for editing. When the file is opened there is small rectangle next to the radio button that changes to cyan to indicate that the file is open.

To open new data files, use the small round radio buttons:

Clicking on the radio buttons along the left side of the form will open a new data file form, change the small rectangle to cyan to indicate that the file is open and make the form the current active file. For some files such as CONT.DAT and TOLER.DAT, default values for the variables will be loaded for your review and editing.

Switching between open data files:

The radio buttons can be used to toggle between active data files. Click on the small round, radio button in front of the open data files (those with a cyan colored rectangle) to switch between open files. The radio button with an inner black circle indicates the current active data file.

2.3 Editing the GUI Data Files

FLO-2D uses a series of data files in ASCII text format. You can use the GUI to edit or develop the ASCII text files. Each data file window in the GUI displays all of the required data input for that file. There are no forms in the GUI for the FPLAIN.DAT and CADPTS.DAT files. These files are very large and generally do not need to be edited. To edit the n-values or

elevations in the FPLAIN.DAT, the GDS or the FLOENVIR processor programs should be used to graphically change these variables. These two files can also be opened in an ASCII editor such as WordPAD. The only time it is necessary to edit the CADPTS.DAT file is when grid elements are added to an existing project. In that case, both the FPLAIN.DAT and CADPTS.DAT files will have to be modified.

Some of the data in the various file forms can be directly edited, but some data boxes require additional pop-up windows. Data entry and editing can only occur in white data boxes. Shaded gray boxes are not accessible for data editing because the appropriate dependency has not been created yet (such as a 'switch' check box). For example, the variables in the SED.DAT file can not accessed without checking the Sediment Transport 'switch' box in the CONT.DAT file. Shaded dialog boxes may also include list boxes and data grid tables. If you see data in a gray blue box, it has been entered in some other location, perhaps from a lower cascading GUI data table. If a row of data or selection in a grid list or data box can not be filled out, then look for a corresponding "Add..." or "Change..." button. The "Add" or "Change" buttons described below will initiate a pop-up window to access the data entry you are trying to edit.

"Add", "Change" and "Delete" buttons:

These buttons enable data editing for grid tables or lists that contain dense data. Buttons with continuing periods such as "Add...", "Change..." or "Delete..." indicate that a 'hidden' pop-up window will be used to edit data. Clicking on these buttons will expose the data editor window. The pop-up windows are self-explanatory and contain white data entry boxes.

Cascading data input:

You may want to enter data, but the table box is grayed out. There are dependencies in some of the data tables that have a nested hierarchy. The topmost table grid in the system displays the highest level of information. When a cell is selected, the entire row is highlighted and the next dialog box is opened with information about the selected row. Similarly, when a cell in the next table is selected, still another list dialog box may be accessed. For example, in the STREET.DAT file, the clicking on the Street Name in the group of dialog boxes label Street Data will load the street data in the middle dialog box. In turn, clicking on Street Node, will load the bottom dialog box of flow directions. Adding a street segment with flow direction and width row in the bottom table will change the number of segments in the middle table. This creates a cascading hierarchy of data in the table boxes.

ASCII text mode data:

The GUI has an ASCII text mode to edit the files as they would appear to be read by the FLO-2D model. On occasion it may be simpler for you to edit the data in the ASCII mode. On the top menu, click on the 'Display' button and a 'Display Text' button appears. Click on this button to switch to the ASCII data file. The data is displayed in the format described in the FLO-2D manual. With the menu Edit button or the right mouse button you can select, cut, copy, paste and undo the data.

Some files such as CHAN.DAT or INFIL.DAT can become very large. When the data files become too large and you want to edit large portions of the file such as entire columns of data, use another ASCII text editor such as WordPad, NotePad, COED (HEC program editor), or TextPad® instead of the GUI. You may also edit the data files by columns or rows in data processing programs such as Excel.

Hint: If you have any problems in editing the data in the GUI, the FLOENVIR program or another editor, before you run the model, scan the data files with an ASCII editor to ensure that the data is correctly formatted. Check the end of each file for any misplaced data, extra lines or inadvertent keystrokes. The Data Input Manual descriptions below govern the data file format in the model. Review the various example project files (e.g. BARN or MONROE) to compare the data file format.

Help comments:

There are two sources of instructional information in the GUI Help pull down menu. The Help Window displays a comment about each variable and its potential range as you move the mouse over the variable name. The original variable name is listed in the Help Window as it appears in the FLO-2D manual. The Comment Window provides some instructional comments regarding the open data file. These comments can also be reviewed in the manual.

2.4 Running the FLO-2D Model from Explorer

The FLO-2D model and all the processor programs are stand alone executable files that can be run from any location in the C-drive of the computer. The FLO.EXE program must be in the project subdirectory with the data files. After the data files have been completed, open and Explorer and locate the project subdirectory. If the FLO.EXE (or processor program) is not in the project subdirectory, copy it from the FLO-2D subdirectory and paste it into the project subdirectory. To run the model, double-click on the FLO.EXE file name in Explorer.

The FLO-2D flood simulation can be terminated at any time during the run by clicking on Exit on the window menu. The simulation will terminate after the current timestep is complete and the output files will all be generated and saved. This enables the user to recognize if the flood simulation is running poorly (e.g. too slow or not conserving volume) and stop the simulation without losing the opportunity to review the output data.

2.5 Starting from Scratch on a New Project

The first task in starting a new project is to obtain a suitable map or a set of digital terrain model (DTM) points that will display or depict the watershed, floodplain or alluvial fan potential flow surface. If the DTM points are not available, it may be necessary to digitize a suitable map. The map can be digitized in a CADD program or in the GDS if an map image can be generated or scanned. If bridges, culverts, buildings and streets are to be simulated, the user has to locate

these features with respect to individual grid elements. Aerial photography or orthotopo maps may be useful for this purpose. It is helpful to produce a hardcopy of the grid system overlaid on the mapping or image for reference while working on the data files.

The GDS is flexible tool that will simplify the process of creating the grid system. It's application is discussed in detail in a later section. If a user has CADD program resources and chooses to generate the topographic files FPLAIN.DAT and CADPTS.DAT with using the GDS, it is only necessary to create a DTM file with x- and y-coordinates and elevation. A CADATA processor program will convert the DTM data file into the required FPLAIN.DAT and CADPTS.DAT files. Instructions for using the CADATA program are located in the appendix of this manual.

Before generating the finite difference grid system in the GDS, it is necessary to select the size or width of the square grid element (Δx) that will provide as much simulation detail as practical without exceeding the resolution of the mapping or DTM points. Generally, the grid element width will range from 50 ft to 1000 ft (15 m to 400 m). A good 'rule of thumb' for selecting a grid size is to divide the highest peak discharge in the all inflow hydrographs by the proposed grid element surface area A_{surf} (e.g. 100 ft x 100 ft) to get a unit discharge (cfs/ft^2 or cms/m^2). The resultant value of Q/A_{surf} should be less than 1.0. The closer that Q/A_{surf} can approach 0.1, the faster the simulation will run.

If project area is relatively small compared to the hydrologic system that needs to be model, more than one FLO-2D simulation could be considered. A coarse grid system can be established for entire 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.

The user must decide whether to simulate the entire rainfall-runoff event FLO-2D or use existing hydrograph data that may have been generated with another hydrologic model such as HEC-1. If the inflow hydrograph already exists, it is suggested that the rainfall/runoff results be carefully reviewed in terms of unit peak discharge and percent loss for regional and local accuracy. It is recommended to use existing flood hydrograph whenever possible. If rainfall-runoff is going to be modeled with FLO-2D, a design storm including total precipitation, duration and distribution must be determined. The user must also decide whether infiltration will be simulated.

Channel flow and street flow can require extensive field data. For channel flow, cross sections should be surveyed. It is not always necessary to survey actual channel elevations; relative elevations for the survey coordinates may be sufficient if the floodplain topography and grid element elevations area relatively accurate. The cross sections can have an arbitrary base point elevation on the left bank (e.g. 100 or 1000). Actual cross section elevations when available will improve the accuracy of the channel-floodplain discharge exchange.

Several guidelines for starting a FLO-2D simulation follow:

- Begin with a basic simulation involving only overland flow. When you have successfully run the overland flow model then add channels, streets, infiltration and other components in subsequent runs turning off previous model component switches. This approach will enable the user to observe the effects of applying the various FLO-2D components on the flood distribution.

- When the data format appears to be confusing, review the provided example project data files.

- It is important to note that output files in the subdirectory will be overwritten during subsequent model runs. There are two choices to saving data that may be overwritten: 1) Rename any output files that should be saved; 2) Create a new subdirectory, copy all the *.DAT files into it and then start a new flood simulation.

- Create separate input hydrograph file for clear water or mudflow simulations. The sediment concentration assigned to the discretized hydrograph has to be removed for a water flood simulation. Create one file INFLOWW.DAT for water and INFLOWM.DAT mudflow, then copy the appropriate file to INFLOW.DAT before running the model.

- If a CADD program is used to assign elevations to the grid elements, sometimes the elevation interpolation for individual grid elements is poor. For example, if a highway on-ramp is located in the middle of a grid element, the on-ramp elevation may be assigned instead of the floodplain surface elevation. The GDS was developed with filters to avoid this problem. The GDS can filter out both high and low elevation DTM data that doesn't represent the floodplain surface and average the remaining DTM points to assign a more accurate floodplain grid element elevation.

- The user can watch the flood progression over the flow surface during model runtime using the graphics display routine. To initiate the graphics display, the user will have to assign a hydrograph to an inflow grid element and define a screen refresh time interval in the CONT.DAT data file. Only one hydrograph can be plotted in the graphics display during the flood simulation. If rainfall is being simulated, the cumulative total precipitation will be plotted automatically.

2.6 Metric Option

The user can choose either the English or Metric system of units (for the Metric system set METRIC = 1 in the CONT.DAT file). When using the Metric system, substitute the appropriate metric unit for the English unit in the data files. The following basic units are used in the model:

Variable	English	Metric
discharge	cfs	m^3/s (cms)
depth	ft	m
infiltration 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
yield stress	dynes/cm^2	dynes/cm^2

2.7 FLO-2D Example Flood Simulations

There are five example projects that can be reviewed. In addition, an example of all the FLO-2D data files (not necessary for the same project) is contained in the EXAMPLE subdirectory. Use FLO-2D GUI, WordPAD or other ASCII editors to examine or edit the example data files. The FLO-2D data files have an *.DAT extension, and the FLO-2D output files have an *.OUT extension. Scan the project subdirectory to identify the various output files after a FLO-2D simulation is complete. Use MAXPLOT or MAPPER to view the flood inundation results.

To run the examples, click on the FLO-2D icon, open the CONT.DAT file in the project subdirectory (e.g. BARN), click on *Execute* in the top menu, and then click on the FLO-2D button. From Explorer, make sure the FLO.EXE file is in the project subdirectory and double click on the file name. The example simulations are setup for the graphics mode and will generally take only a few minutes to run depending on your computer capabilities. The graphics mode displays a plot of the flood hydrograph and a view of the floodwave progression over the grid surface while the model is running. You can experiment by altering the data files (e.g. changing a mudflow simulation to a water simulation, turning off the channel or street options, adding infiltration, etc.). These example data files can be used to create your own project files by copying them to another subdirectory and writing directly over the existing data.

Mudflow Simulation - Barnard Creek, Utah (BARN subdirectory)

An example mudflow simulation is provided for an urbanized alluvial fan (Barnard Creek) near Centerville, Utah. This model simulates a mudflow debouching from a small watershed ravine onto a very steep alluvial fan with numerous streets and buildings. The

potential flow surface consists of 521 grid elements (100 ft square). The mudflow enters the grid system at element 486 (a debris basin), flows down a steep street and spreads out into the residential area. The mudflow is viewed overflowing the street, entering side streets and developed lots and becoming more fluid as the floodwave progresses downslope. Buildings have been simulated to account for the loss of storage and flow redirection. The mudflow simulation includes variable sediment concentration and the computation of viscosity and yield stresses.

Alluvial Fan Channel Overbank Water Flooding - Monroe Creek, Utah (MONROE subdirectory)

The second example flood simulation depicts channel-floodplain discharge exchange on a large, undeveloped alluvial fan that experiences primarily water flooding. The grid system consists of 3,026 grid elements, 200 ft on a side. The flood hydrograph has a peak discharge of 1,900 cfs. The channel is represented by 47 channel elements with variable channel geometries; rectangular, trapezoidal and natural-shaped. The flow is confined along the channel by berms that have been represented by higher grid element elevations. The overbank flooding can be observed returning to the channel.

Desert Southwest Alluvial Fan Flooding - Whiskey Pete's, Nevada (WHISK subdirectory)

Whiskey Pete's is a classical, undeveloped, conically shaped desert alluvial fan. Multiple distributary channels and infiltration are simulated. The hydrograph peak discharge is in excess of 13,000 cfs. This flood simulation is a good project to compare results with the FEMA Fan Model. The conical shape is nearly perfect for a desert fan and there is no development or channelization. The results show that the FEMA fan model underpredicts the flow hydraulics near the fan apex and below the mid-point of the alluvial fan.

Rainfall-Runoff - Blue Diamond Watershed, Las Vegas, Nevada (Diamond subdirectory)

Rainfall, infiltration and runoff are simulated for a 34 square mile desert watershed with 10,472 grid elements 300 ft square. Steep, unvegetated ridges, bedrock and numerous small watershed ravines characterize the area. The drainage pattern can be observed as the rainfall-runoff develops in the graphics mode. The higher elevations and ridges drain first and the water collects in the various ravines and gullies.

Large River Flood Inundation - Truckee River, Reno Nevada (TRUCK subdirectory)

In January 1997, the Truckee River flooded a large area including portions of the Cities of Reno and Sparks, Nevada. FLO-2D was selected by the Corps of Engineers to replicate the 1997 flood in this example project. Over 100 channel cross sections were surveyed to simulate the overbank flooding. Bridges, some streets and buildings, infiltration and levees are simulated. Partial bridge blockage is simulated with a doubled pier width. The flood inundation covers a large, flat area that requires a long time to drain. There are three tributaries in this reach, each with an inflow hydrograph.

2.8 Exporting Topographic Data from CADD Programs

The grid developer system (GDS) will facilitate the preparation of the grid system. It will overlay the grid system on a digital terrain map or a DTM set of points, assign elevations to the grid elements and will automatically prepare the FPLAIN.DAT and CADPTS.DAT files. The use of the GDS eliminates the need for a CADD program and the application of the CADATA processor program. The GDS is described in the following section.

As an alternative approach to creating the grid system and assigning elevations with the GDS, a computer aided design and drafting program (CADD) can be applied. Most CADD software programs with a digital terrain model will be able to create a grid system and interpolate elevations. The objective of using the CADD is to export to a file the grid system data of the potential flow surface from a digitized map in the following format:

Grid Number	X-Coordinate	Y-Coordinate	Elevation	Manning's n

Starting with a digitized map in the CADD program, the random survey grid routine can be used to overlay a system of square grid elements. With a DTM model, elevations can be interpolated from the grid element nodal points. The grid element coordinate and elevation data can then be exported to an ASCII file. Using this data file and the CADATA processor program, the FPLAIN.DAT and CADPTS.DAT files are created.

III. PRE-PROCESSOR PROGRAMS

3.1 Introduction

There are seven pre-processor programs to help to create or edit the FLO-2D data files: FLOENVIR, GDS, PROFILES, CADATA, CHECKER, POWER, and XSEC. For FLO-2D Version 2003.06, only the GDS, FLOENVIR and PROFILES program are still necessary to develop a flood routing model. The rest of the processor programs are obsolete, but are provided in the FLO-2D subdirectory, in the case that there may an occasion to use them. A detail discussion of the GDS, FLOENVIR and PROFILES program is included in this manual. A brief description of the obsolete programs follow with a more complete set of instructions in the Appendix.

- *CADATA* is a preprocessor program that reformats the topographic grid point data exported from the CADD program DTM into a data file used by FLO-2D. The CADATA program reorganizes the topographic data into a file format that identifies contiguous elements in the four directions. The GDS made the CADATA program obsolete.

- *CHECKER* is an old DOS program that will check the FPLAIN.DAT file for errors in contiguous grid elements. A grid element (say 224) that has a neighbor element to north that is 256 must be the neighbor of grid element 256 to the south. These grid element neighbors must be cross correlated in the FPLAIN.DAT file. *CHECKER* will report on any errors in the FPLAIN.DAT grid element organization. If grid elements have been added to the FPLAIN.DAT file, *CHECKER* can be used to check the accuracy of the modifications to the file.

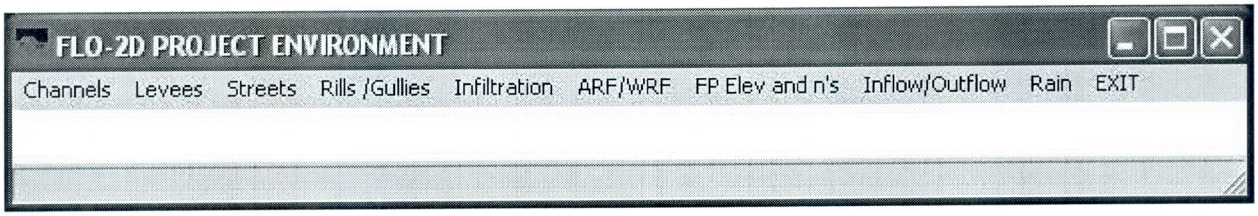
- *POWER* is a simple regression program that will determine a power regression relationship of the form $Y = a X^b$ between any two variables and report the statistics of the least squares fit analysis. This program may be useful to determine a bridge or culvert rating curve.

- *XSEC* computes the channel geometry relationships as a power function of the depth from the cross section survey data base. The channel geometry relationships include $A = a d^b$, $P_w = a d^b$, and $T_w = a d^b$, where A = flow area, P_w = wetted perimeter, and T_w = channel top width and a and b are a regression coefficient and exponent respectively. The FLO-2D model can use these channel geometry relationships to perform the flood routing as one option. It is suggested, however, that if the cross section survey data is available, the cross section channel routing option should be used instead.

3.2 FLOENVIR

A graphical working environment FLOENVIR was created to facilitate editing the grid element attributes. The FLOENVIR displays the grid system and provides tools to zoom, pan and view extents. Grid elements can be selected individually or as groups by single selection with the mouse, painting multiple grids with the mouse or drawing a polygon around groups of grid elements with the mouse. With the FLOENVIR program, you can edit or add channels, levees, streets, rills and gullies, infiltration, ARF/WRF values, the floodplain elevations and roughness and the inflow and outflow nodes.

To open the FLOENVIR program, click on FLOENVIR from the GUI pre-processor pull down menu or put the FLOENVIR.EXE file into the project subdirectory and double click on it in Explorer. The main window displays a series of menu items.



Click on any component menu item to display the grid system.

To start editing any of the grid element attributes, zoom in on portion of the grid system using the zoom icon on the tool bar (see toolbar image pg. 16). Popup dialog boxes, instructional boxes and the selection icons will provide you with guidance to edit the data. The FLOENVIR will greatly facilitate assigning spatially variability to grid element attributes. It will also help you visualize constructing a channel, levee and street networks. FLOENVIR saves the data in the correct ASCII format for the model. Save the data often and create a backup copy of your data files before using the FLOENVIR.

The steps used in the FLOENVIR program are:

1. Create the grid system (FPLAIN.DAT and CADPTS.DAT) and complete the CONT.DAT, TOLER.DAT, INFLOW.DAT and OUTFLOW.DAT prior to starting FLOENVIR. You can just enter zeros for the required lines in INFLOW.DAT and OUTFLOW.DAT. For example, it is not necessary to enter the outflow nodes before using FLOENVIR. The assignment of the outflow nodes can be done in FLOENVIR. You can guess at the parameters you are not sure of in the initial setup of these files. If you are just starting a new project, turn off all the component parameter switches in the CONT.DAT file.

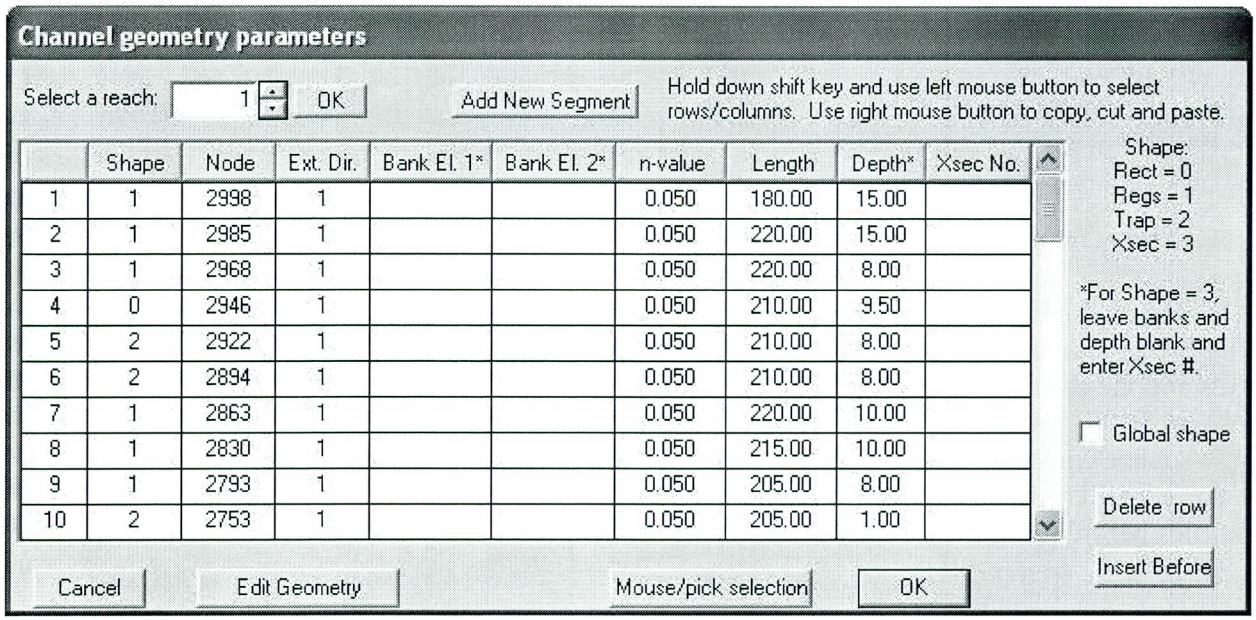
2. Use the GUI to access the FLOENVIR or put the FLOENVIR in your project subdirectory and double click on it in Explorer.

3. After you open FLOENVIR, click on a menu item such as channels as shown above.

4. Zoom in on a portion of the grid system. ZOOM in until you can read the grid element numbers. Use the various toolbar commands to zoom, pan and view extents to move around on the grid system.
5. Select a method for editing from the toolbar (last 3 icons). You can choose Pick, Polygon or Mouse (painting). For some component attribute editing, only 1 or 2 selection methods may be available.

6. When using the editor tables, dialog boxes or a message at the bottom of the screen will usually lead to the next step. Most of the dialog boxes and tables are straightforward and intuitive. Using channels as an example:

    a. ZOOM-IN on a portion of the floodplain and select Edit Channel Geometry. A dialog box will appear.



	Shape	Node	Ext. Dir.	Bank El. 1*	Bank El. 2*	n-value	Length	Depth*	Xsec No.
1	1	2998	1			0.050	180.00	15.00	
2	1	2985	1			0.050	220.00	15.00	
3	1	2968	1			0.050	220.00	8.00	
4	0	2946	1			0.050	210.00	9.50	
5	2	2922	1			0.050	210.00	8.00	
6	2	2894	1			0.050	210.00	8.00	
7	1	2863	1			0.050	220.00	10.00	
8	1	2830	1			0.050	215.00	10.00	
9	1	2793	1			0.050	205.00	8.00	
10	2	2753	1			0.050	205.00	1.00	

    b. Choose the Mouse/pick selection button at the bottom of the box.

    c. Select Mouse (paint) edit tool bar icon.

    d. Pick the grid elements until you are finished with the available channel elements on the ZOOM screen.

    e. Select 'OK' on the small dialog box and it will return you to the 'Channel geometry parameters' dialog box.

    f. Locate the row in the first column that you want to insert the selected channel elements.

    g. Click on the INSERT BEFORE button (see above).

h. Fill in the rest of the 'Channel geometry parameters' dialog box columns that are appropriate.

i. Click 'OK' and another dialog box will appear that has the remaining channel geometry for the type of channel cross section you have selected.

j. When you are finished and have clicked 'OK', you will be returned to the Zoom screen. Now you can PAN around or ZOOM EXTENTS or reZOOM, then you can choose MOUSE edit again and continue selecting grid elements.

Note: When you have several channel elements started in a reach, choose 'EDIT CHANNEL CONTROL' and 'EDIT SEGMENT CONTROL' to fill in the control lines in the CHAN.DAT file. You need to EDIT 'CHANNEL CONTROL' only once for the CHAN.DAT file. You need to 'EDIT SEGMENT CONTROL' for every segment reach.

7. When you save the files, the appropriate component switch will be turned on.

The FLOENVIR toolbar has the following icons (in the order of appearance from left to right):

a. Save the open data file.

b. Exit and close the FLOENVIR program.

c. Print the screen.

d. ZOOM-IN on a portion of the grid system.

e. Pan or move around in the zoomed view.

f. Zoom extents or zoom out to the full view of the grid system.

g. Pick an individual grid element.

h. Draw a polygon around a group of grid elements with the mouse.

i. Paint a group of grid elements by holding the left mouse button down and moving the mouse around.

8. Click on Return in the top menu to access the other components.

9. The other model components can be modified in a similar manner. The important steps to remember are:

a. ZOOM-IN before editing.

b. After Zooming-In select either a menu editing command or one grid element selection mode (one of the last three icons 'Pick', 'Polygon', or 'Mouse Paint'.

c. Pan around in the zoomed view to select other grid elements.

d. Click on 'Exit' or 'Return' if you have problems.

e. Save your edited data frequently.

3.3 GDS

The grid developer system (GDS) was developed to facilitate the preparation of the grid system. It will overlay the grid system on a digital terrain map or a DTM set of points and interpolate and assign elevations to the grid elements. It will also assign the roughness n-values to the grid elements. The GDS will automatically prepare the FLO-2D model FPLAIN.DAT and CADPTS.DAT files. Using GDS eliminates the need for a CADD program to develop the topographic data for the FLO-2D model.

3.3.1 GDS System Overview

The flow domain complex topography represented in the FLO-2D model can be quickly developed using GDS interactive tools to establish geographical boundaries, input background maps or DTM point files or analyze site characteristics. The GDS output files are the FPLAIN.DAT and CADPTS.DAT files that represent the topography and coordinates of the potential flow domain. These files are in a final form for the FLO-2D model error free and require no further attention unless grid elements are added or deleted from the system. GDS utilizes the capabilities of the MS-Windows™ operating system. The computational grid is constructed interactively using mouse point and click events. All data is thoroughly validated in order to minimize common input errors. GDS supports both the English and the SI system of units.

The basic procedure is to import the DTM points (x-, y-coordinate and elevation ASCII format) from aerial photography or LIDAR into the GDS and overlay a square grid system. The grid system can be trimmed to fit the potential flow area. A computational domain surface inside the grid system boundary is then identified and the grid elements are assigned elevations. The GPS interpolation routine for assigning grid elements is superior to that in a CADD program. The GDS examines the random elevation points within a user specified radius of every grid element center. A selected filter routine then identifies those elevation points that are significantly higher than the floodplain and eliminates them from consideration in calculating the elevation. The elevation of the grid element is then computed using an inverse weighted distance average scheme.

The GDS system includes the following components:

- ESRI shape file format data such as land use, soil types, and Manning roughness coefficients can be imported.
- ESRI ArcInfo ASCII grid files containing terrain elevations and NOAA rainfall isopluvial data can be imported.
- Multiple geo-referenced aerial photos in various graphic formats such as TIFF, BMP, JPG, etc. can be imported and created as background.
- Multiple ESRI ArcInfo ASCII grid files can be listed in a tile and index catalog file and referenced to a user defined polygon in the study.
- Multiple image files like aerial photos can be listed in a tile and index catalog file and referenced to a user defined polygon in the study area.
- New multiple layer capability is available including extensive control of layer properties.
- Zoom and pan features enhance views.
- Green-Ampt infiltration parameters based on the Maricopa County, Arizona Drainage Design Manual can be computed.
- Spatially variable Green-Ampt infiltration parameters can be assigned to FLO-2D grid elements based on soil shape files, land use shape files, and soil and land use properties tables.
- Spatially variable Manning roughness coefficients can be assigned to FLO-2D grid elements based on Manning shape files.
- Spatially and time variable rainfall data can be computed and assigned to FLO-2D grid elements based on multiple NOAA rain data files.
- Channel cross sections can be cut from regular or irregular elevation points and assigned to particular channel segments.
- Cross section data in HEC-RAS G-format can be converted to the FLO-2D channel data format.
- Channel segments can be created and edited.
- HEC-1 hydrograph can be imported and managed.
- Automatic selection of computation area from user defined polygons.
- DTM points can be deleted from the data base to improve elevation interpolation.

The GDS program has been completely redesigned to incorporate ESRI MapObjects© GIS software controls version 2.1. A discussion of all the various tools and components related to ESRI and GIS interface is presented in Section 4.

3.3.2 Getting Started on the Grid Developer System

Installation Requirements

The GDS was installed in the FLO-2D subdirectory during the installation of the FLO-2D program. No further installation is required. The GDS is subject to the same computer system requirements as described for the FLO-2D program in section 1.2.

IMPORTANT NOTE: *For the* GDS *system to function properly, the MS-Windows™ environment must be set to a decimal point number format using the International (Regional) option of the Control Panel.*

3.3.3 Installation of Other Required DLL files:

GDS uses ESRI MapObjects© GIS software controls version 2.1. These components require several data base libraries that may not exist in all computers. In order to ensure that all required DLL components exist, do the following:.

1. Copy the following files to your computer: MO21rt.EXE and DAO35.exe. You may find these files in the distribution CD.
2. Run these programs with the corresponding arguments as follows:

   MO21rt.EXE /FGHMJ and DAO35.exe /d=ABCDF /ni
3. Restart your computer.

IMPORTANT NOTE: *In some computer systems it will be required to install other Microsoft data DLL components. To verify if you have these DLL components installed, run the MDAC_TYP.EXE program and follow the instructions to register all files. Further directions to apply the MDAC_TYP.EXE can be found at:*

http://www.microsoft.com/data/mdac27info/en/readme.htm

3.3.4 Data Requirements

GDS either requires or uses the following data:

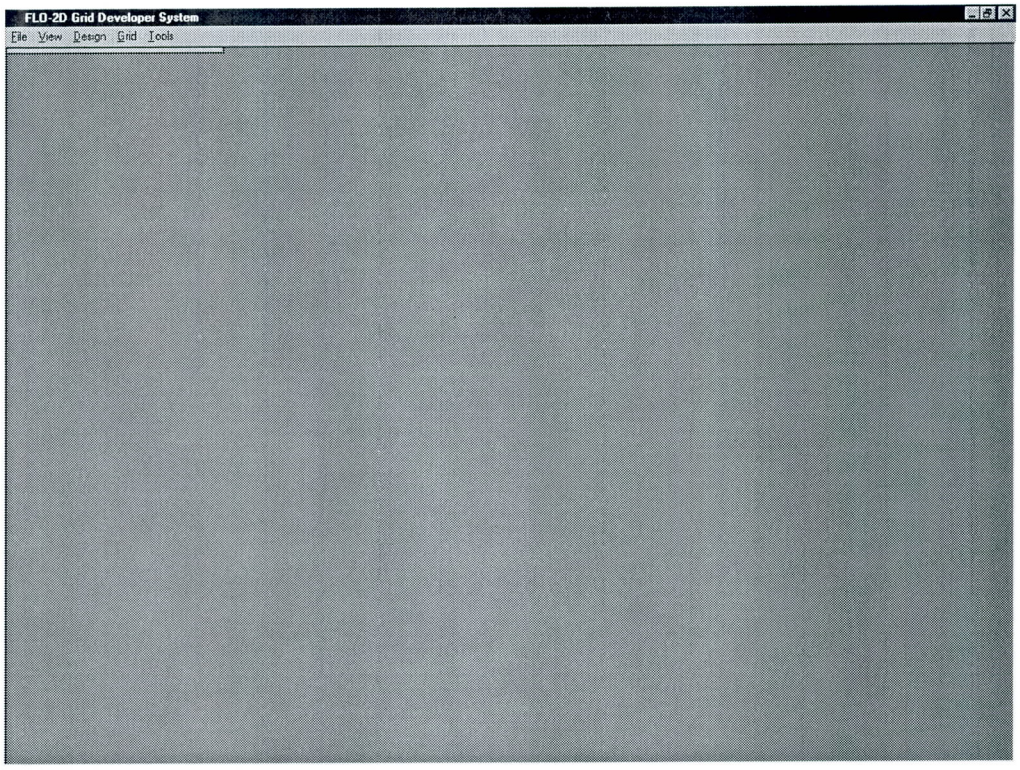
- Terrain elevations given as random, irregular points (DTM);
- Terrain elevation given in ArcInfo ASCII grid files;
- Study region limits;
- Manning's roughness coefficients shape files;
- Soil shape files;

- Land use shape files;
- Soil type tables;
- Land use tables;
- Image files in the following formats: BMP, JPG, ArcInfo INFO Grid, GeoTIFF TIFF with a Geo header, Image Catalogs, JPEG, MrSID, TIFF, JPEG, Bmp.;
- NOAA rainfall data given in ArcInfo ASCII grid files.

These data are introduced through window dialog boxes and may be stored in files for later retrieval. User's input is validated to avoid out-of-range values. Default or recommended values are presented when the dialog boxes open.

3.3.5 Developing the Grid System

GDS will create modeling flow domain in an area of the screen called *working area or region*. When you double-click on the GDS icon, the start window looks like this:



.

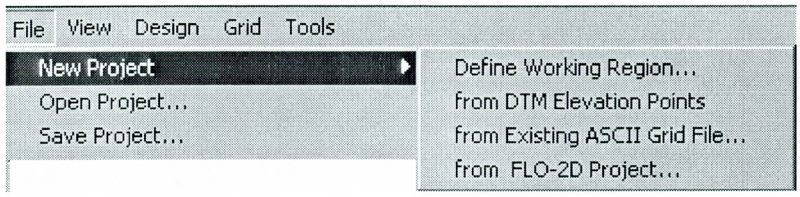
To create a new grid system, you will be performing the following steps:

1. Create a working area with the particular region limits of your project (this may include importing the DTM points);
2. Design the geographical boundaries of the project site or flow domain;
3. Input the terrain data or interactively generate topographic contours (you may input the DTM data at the outset);
4. Create the base grid system;
5. Select the simulation area (computational domain) for the FLO-2D model;
6. Set Manning's roughness coefficients (n-values);
7. Interpolate the digital terrain elevation data and assign FLO-2D grid element elevations;
8. Create FPLAIN.DAT and CADPTS.DAT, FLO-2D data files.

You may also:

- Calculate Green-Ampt infiltration parameters and update the INIFIL.DAT file;
- Calculate variable rainfall and update the RAIN.DAT and RAINCELL.DAT files.

There are four ways to initiate the GDS program to create a FLO-2D grid system. Begin in the New Project menu:



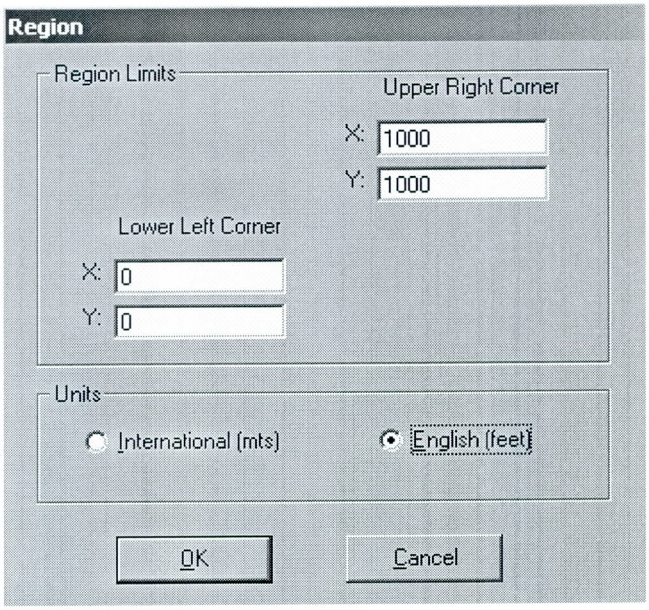
Only **one** of the following four methods is required:

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

These four options are mutually exclusive. Choose only one method to create your project. The first method is analogous to digitizing a map in a CADD program. The GDS program organizes and uses data in layers. When you import data from external files or add data from dialog windows, the system creates new layers for each data category.

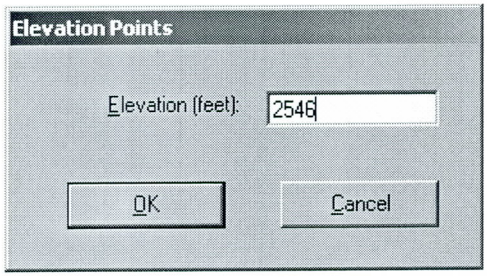
3.3.6 Create a Grid System by Digitizing Points

In this mode you will define the potential flow domain coordinate limits and establish the screen working area. On the *File* menu, point to *New Project* and click *Define working region.* Enter the coordinates of the upper right corner and the lower left corner in the dialog box as shown:



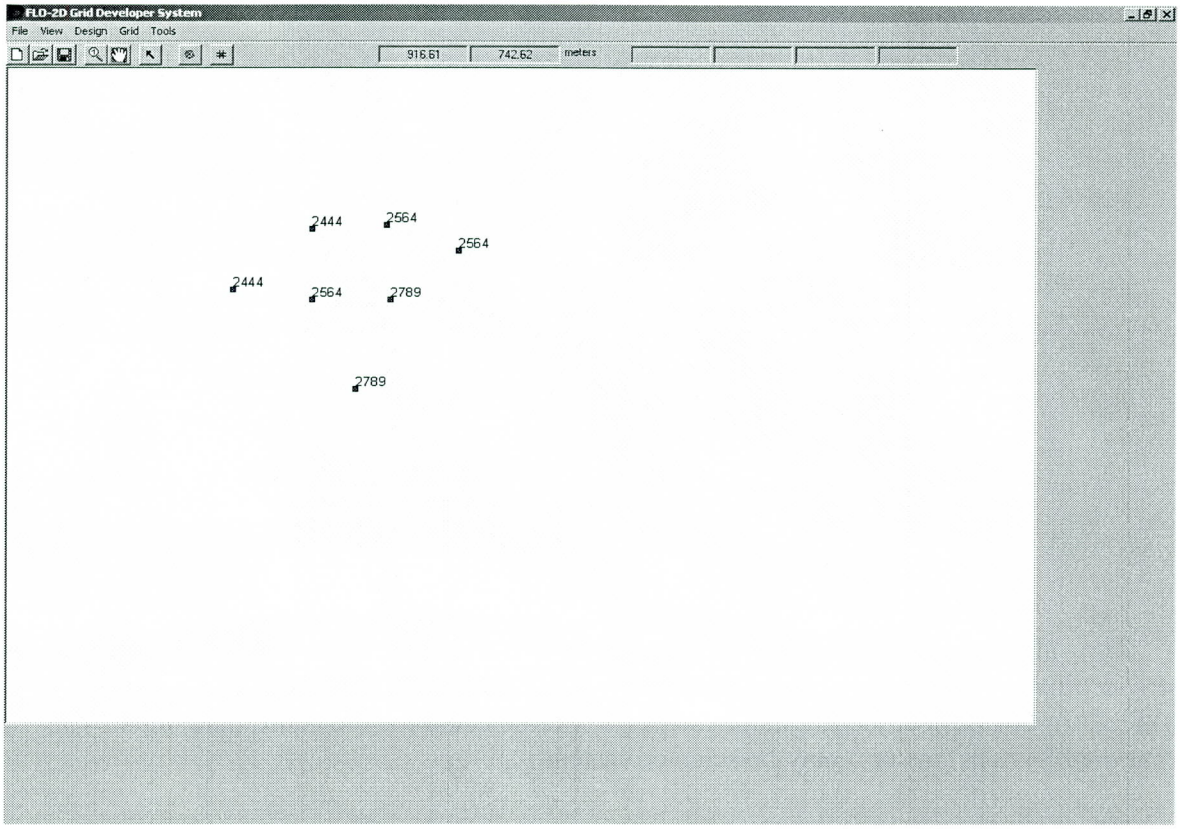
The values entered in the dialog box indicate that the lower left hand corner corresponds to the coordinates (0,0) feet and the upper right hand corner represents the coordinates (1000, 1000) feet. In this project the English System (ft) was selected for the coordinate units. The International System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

When you are finished entering the coordinates, click the 'OK' button. Now you can add random elevation points to the working area. In the toolbar just below the main menu is a set of icons that allows quick access to frequently used commands. To insert a new elevation point, click on the toolbar button or select the Elevation Point/*Insert* command from the Design menu. A dialog box will appear where you can enter the elevation for the points that you are going to define in the working area:



Press 'OK' and then click with the mouse on the point(s) corresponding to this elevation within

the flow domain area. Every time you click the mouse button the point is marked on the screen. Repeat this process for all points with a similar elevation. Since the elevation caption text may increase the display time for large projects, initially captions are not displayed. To add captions or change point symbols you may use the Layer Management tool discussed later.



Please note that the boxes at the right side of Toolbar will indicate the X and Y coordinates of the cursor location in the working area, the Grid Element Index, the Grid Element Number, the Grid Element Interpolated Elevation and the Elevation of the DTM point. You can use a scanned map as a reference image in any of the prescribed formats to locate the points or to digitize contour lines. If the image has contour lines, the GDS does not recognize the contours until the elevation data is entered. You can assign the elevations by following the contour lines and clicking to establish a point elevation at appropriate intervals.

3.3.7 Create a Grid System from DTM Points

The second option to create elevation data in the GDS is to import an existing file of DTM points. This is the most common method for starting a FLO-2D project. The DTM data is generally in ASCII file format listing the coordinates and elevation of each point:

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

The first two columns are the x- and y- coordinates and the third column is elevation (ft or m). To import the file, click the *New Project/From DTM Elevation Points* on the *File* menu and chose the file name. GDS searches for a default filename with *.PTS extension. You must rename the file with this extension. Once the points are read from the file into the program, the points will be shown on the screen and the modeling region will be automatically scaled from the minimum and maximum coordinates. After the DTM points have been imported, the grid system can be overlaid and the computational domain area can be created.

3.3.8 Create a Grid System from ArcInfo ASCII Grid File

Using this option, a new project is created by importing the terrain elevation points from an existing ArcInfo ASCII grid file. Please remember that you can only have one project open at a time. If there is a project already open when you select this command, GDS prompts you to save the current project. The working area is automatically scaled from the maximum and minimum point coordinates. To import the ArcInfo ASCII Grid File (*.ASC) file click on this command in the *File* menu and chose the appropriate filename in your project subdirectory. The format of these files is as follows:

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

.

.

znrows1 znrows2 znrows3 ... znrowsncols /* values of last row*/

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

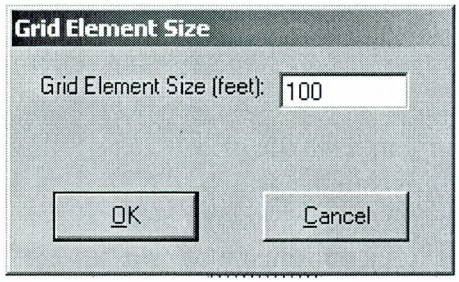
ncols 388
nrows 461
xllcorner 674070.85270015
yllcorner 1000118.1562353
cellsize 100
NODATA_value -9999
2477.259 2480.868 2486.877 2486.877 2487.308 2490.641 2493.438 2493.438
2493.438. . .

3.3.9 Create a Grid System from an Existing FLO-2D Project

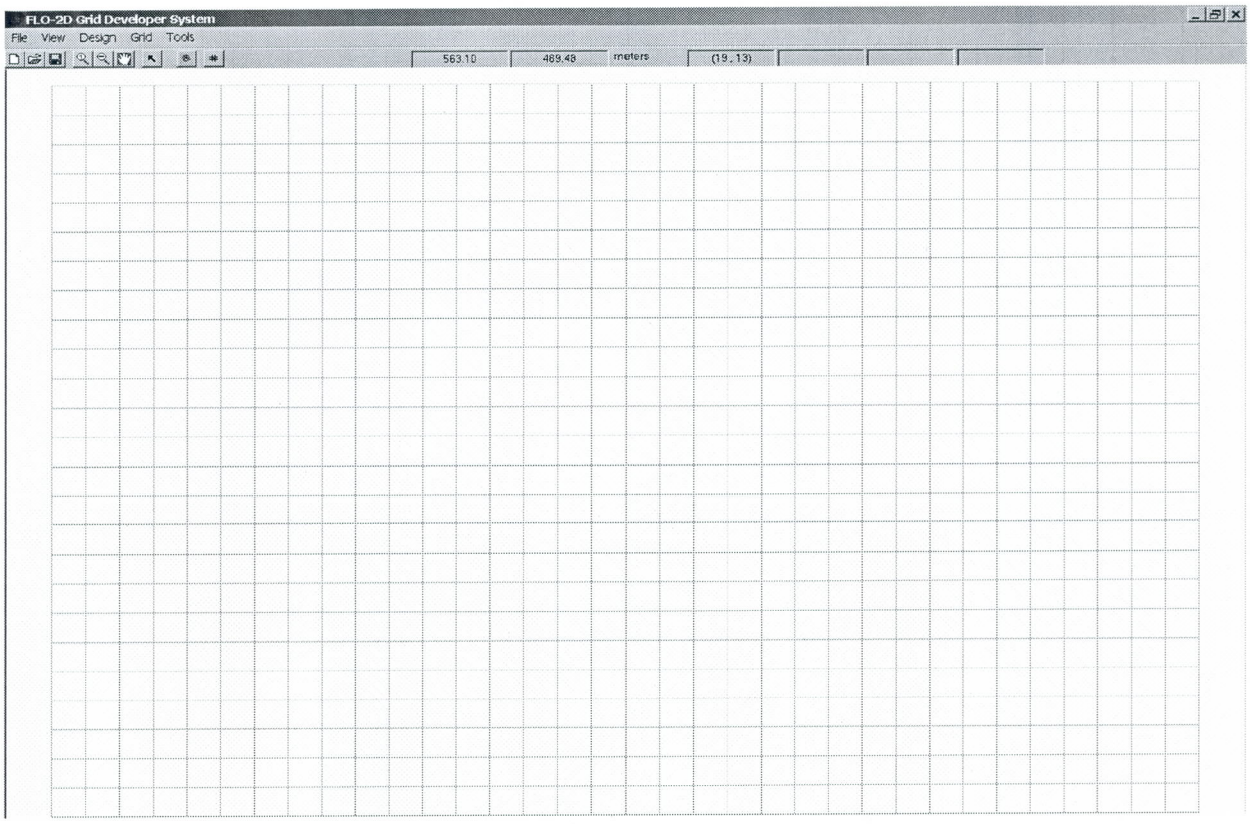
With this option a new project can be created by importing existing FLO-2D data files. Again, you can only have one project open at a time. The working region and the FLO-2D grid system are automatically scaled from the existing project coordinates. To use this option click on this command in the *File* menu and chose the FPLAIN.DAT file from your project subdirectory. The FLO-2D CONT.DAT data file is also required to open an existing project.

3.3.10 Grid System Generation

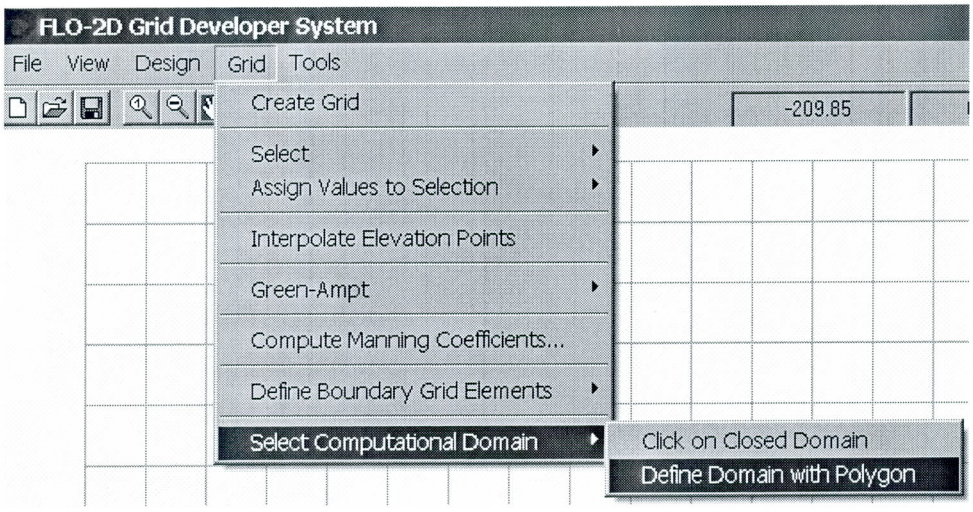
To create a FLO-2D grid system, click the *Create Grid* command on the *Grid* menu and input the appropriate grid element size (square grid) for the project. This size should reflect the resolution accuracy of the topographic data and an acceptable total number of grid elements.



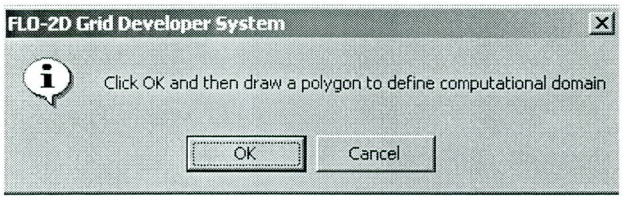
A smaller grid element size increases the FLO-2D model resolution but also increases the computer runtime and memory requirements. A balance should be sought between the grid element size (number of grid elements), resolution of the FLO-2D results and the model computer runtime. Engineering judgment should be used to select the best possible grid size. The following figure displays a system of 50 meters square grid elements.



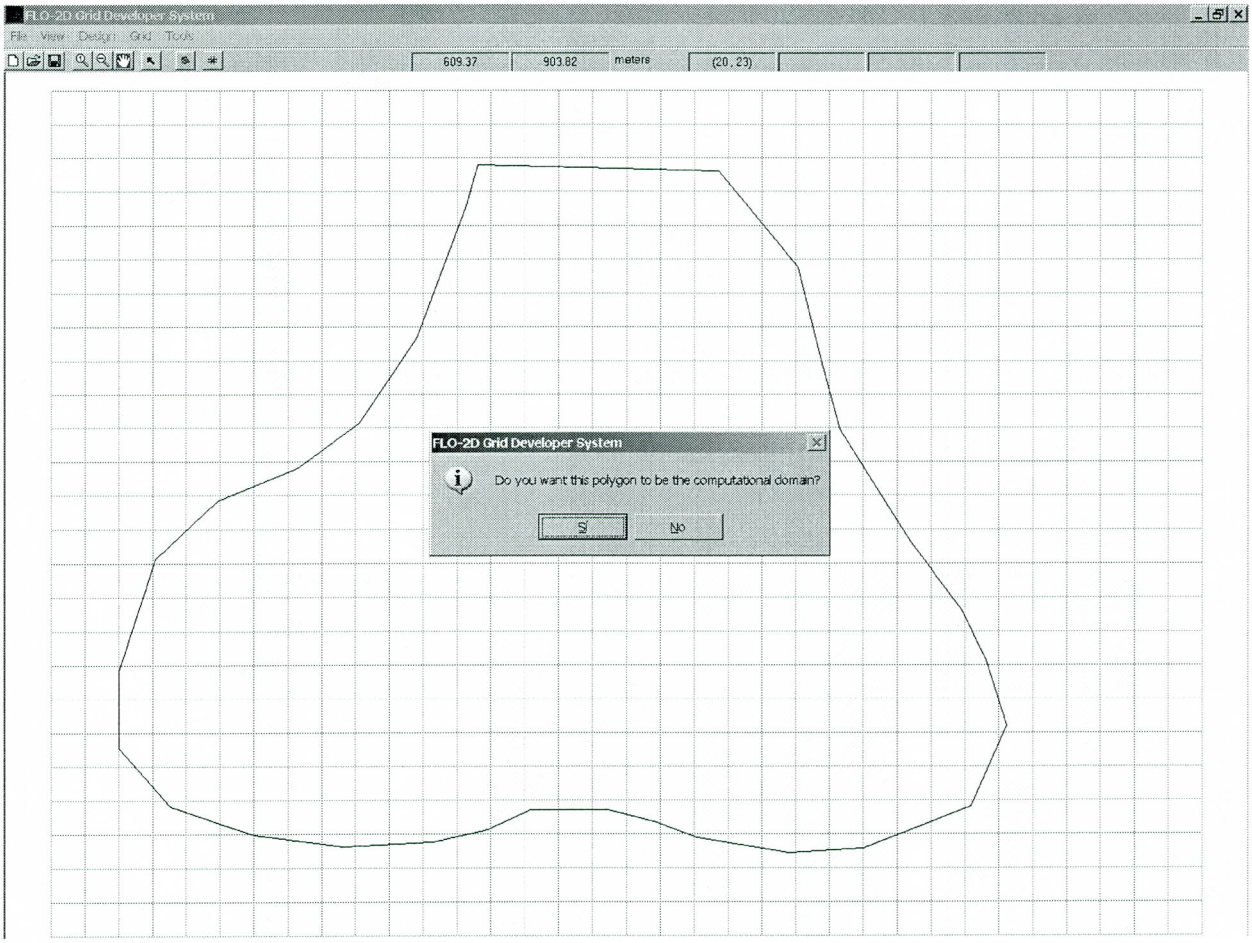
Now it is necessary to define the computational domain by marking the boundary grid elements. The grid system size can be reduced and shaped to accommodate your project area. The grid boundary can be defined by clicking on the *Define Domain with Polygon* option of the *Select Computational Domain* in the *Grid* menu:



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

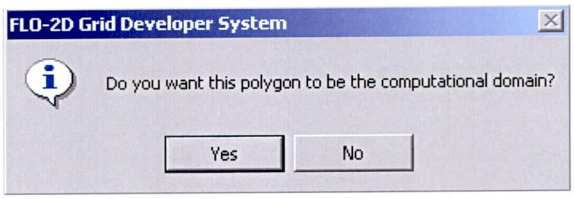


Then the user draws the boundary polygon:

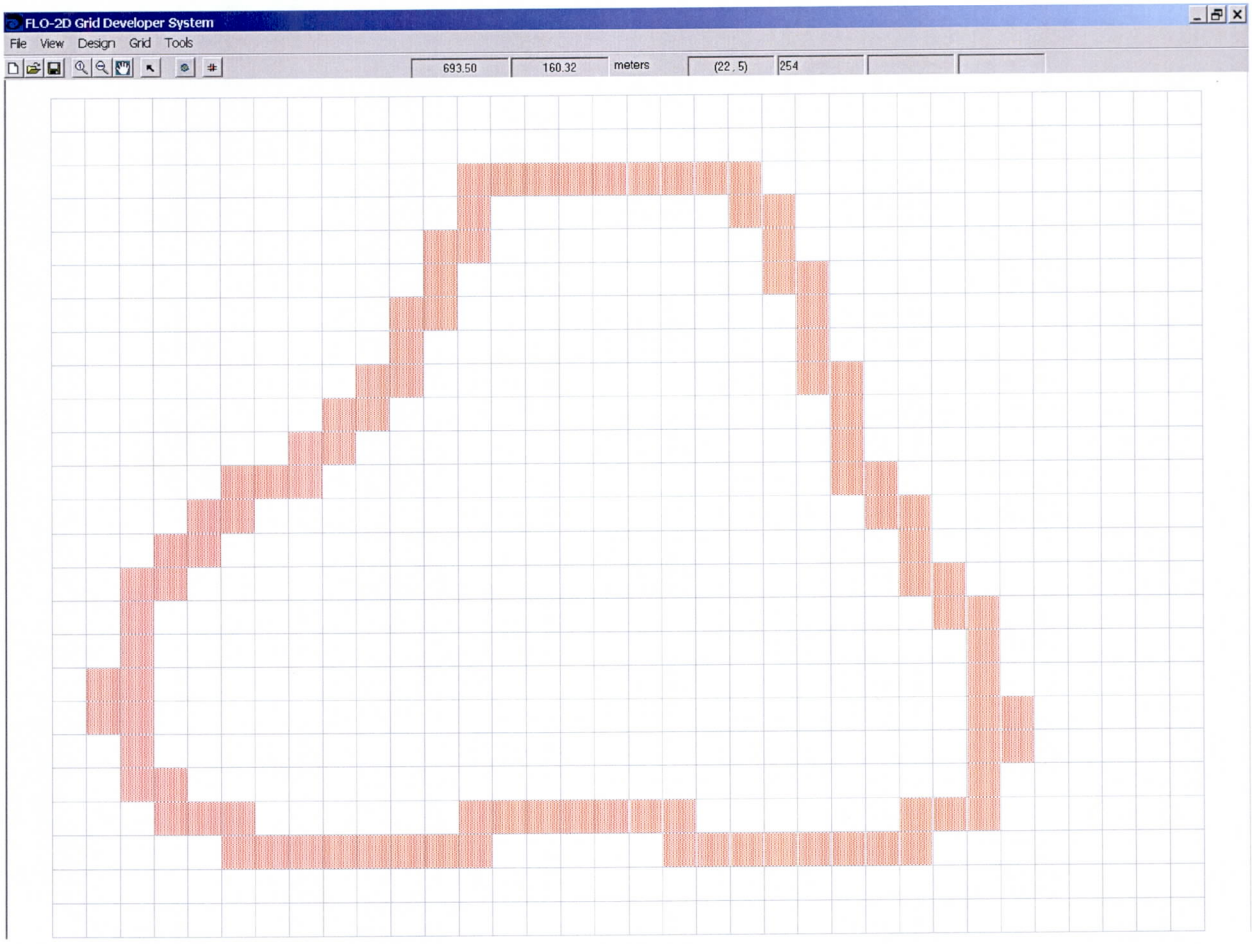


The polygon is completed by double clicking on the last polygon vertex before closing the polygon.

When the user releases the mouse button, the system displays the message:



Answering 'Yes', will create the computation domain while 'No' will erase the polygon. Importing a background image or having the DTM points displayed will enable you to visualize the computation domain and polygon boundary.



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

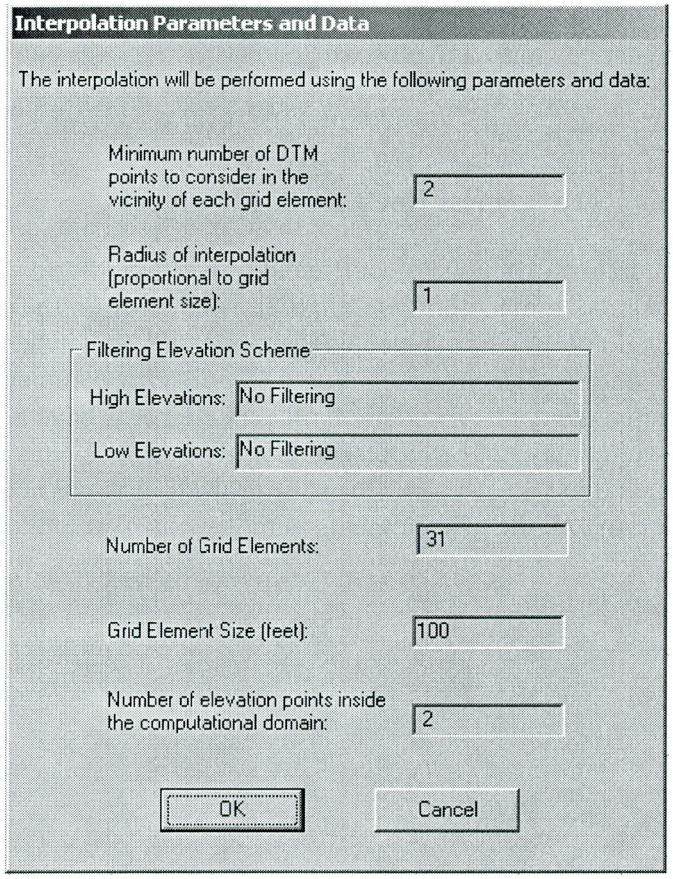
You may mark boundary elements one at time or by painting a line of elements by dragging the mouse while pressing the **shift key**. You can adjust the grid boundary elements by adding or deleting them using the mouse. The boundary elements can be doubled-up or overlap. The objective is to have a completely enclosed flow domain. The boundary grid elements serve no other purpose than to define the FLO-2D computational domain. They have no function in the FLO-2D flood simulation. When you are finished the computational domain will be the area inside the boundary grid elements. Please note that the computational domain is entirely enclosed by boundary grid elements creating a closed region. Click the *Select Computational Domain* command from the Grid menu and then use the mouse to click on any point inside the region. If the area is closed, the program responds with an appropriate message. If this enclosure is incomplete, GDS will inform the user that the area is still open and then additional boundary grid elements must be marked to close it.

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

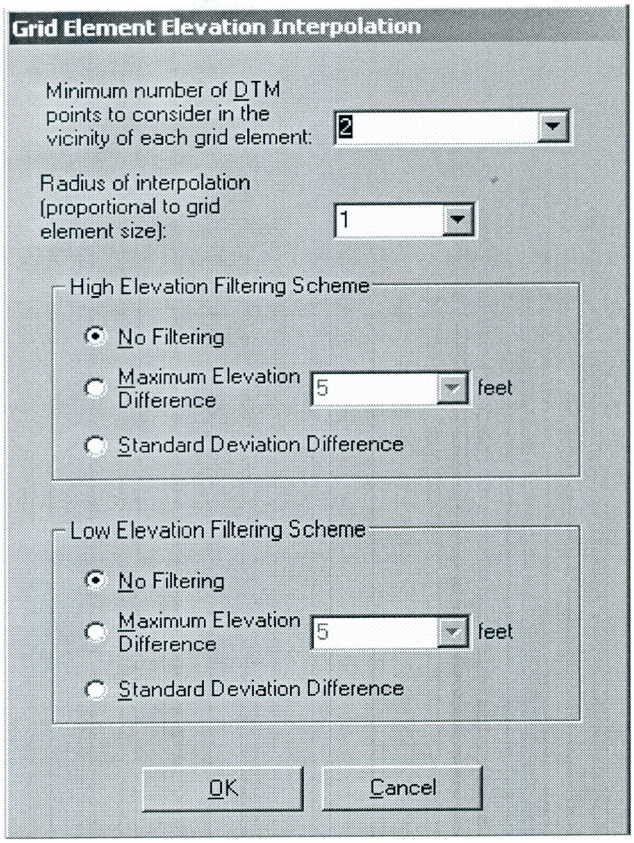
You may input Manning's roughness coefficient to a group of grid elements. To do this, click the *Select/Grid Element* command on the *Grid* menu and click on the elements where you want to define the Manning's coefficient. Selected elements will be hatched diagonally. Now choose the *Grid/Assign Values to Selection/Manning's Coefficients* command and input the desired value for the selected elements. By default all grid elements have a roughness coefficient of 0.040. This default n-value may not be representative of your physical system and you may need to spatially select and adjust the n-values for your project. It is possible to assign the n-values universally by selecting all the grid elements with the *Inner* option under the *Select/Grid Element* command.

3.3.12 Interpolation of Terrain Elevations

A set of DTM points or contour elevations are usually randomly spaced over the flow domain. The FLO-2D model requires that each grid element (center of the grid element) be assigned a representative elevation. To interpolate and assign the grid element elevations from the random DTM points or contour point elevations, click the *Interpolate Elevation Points* command on the *Grid* menu. A dialog box will appear to specify the elevation interpolation parameters:



The user can adjust the interpolation criteria in the above dialog box by clicking *Option/Interpolation* command on the *Tools* menu. A second dialog box will appear that provides the opportunity to control interpolation process.



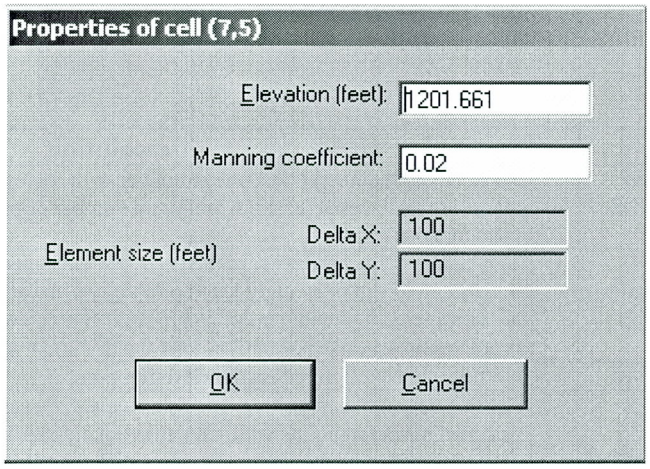
The following table describes the various options for the interpolation algorithm:

Option	Description
Minimum number of DTM points to consider in the vicinity of each grid element	To calculate the interpolated elevation that will represent each grid element, the algorithm analyzes at least the minimum number of points closest to the grid element node.
Radius of interpolation (proportional to grid element size)	This radius defines a circle around each grid element node based on a multiple of the grid elements side. Only DTM or assigned elevation points inside this radius are considered in the interpolation process. If no elevations point exists, the circle is automatically enlarged until at least the minimum number of points is found.
No filtering (High or Low Elevations)	No filtering is performed on the elevation points.
Maximum elevation difference (High or Low Elevations)	For this option, the algorithm calculates a mean elevation using all the DTM and assigned elevation points within the interpolation radius. Then all those points that exceed the selected maximum elevation difference higher (or lower in the case of Low Elevation filter) than the mean are discarded and the mean elevation is recomputed and assigned to the grid element.
Standard deviation difference (High or Low Elevations)	The interpolation algorithm calculates the standard deviation of DTM point elevations and then neglects all those points whose elevations are higher (or lower in the case of the Low Elevation filter) than the standard deviation to determine the mean elevation to represent the grid element.

The interpolation algorithm uses extensive computer resources. The computational procedure should take a reasonable period of time (minutes not hours) depending on the number of grid elements. If there is a large number of grid elements and DTM points, it may require more than 30 minutes to assign elevations to every grid element.

The purpose of the grid element elevation interpolation is to eliminate the need for using a CADD program to assign elevations to the grid elements. The high elevation filtering scheme has been designed to give greater weight to the floodplain elevations in computing a representative grid element elevation. Typically, CADD systems will assign a random survey point elevation to a grid element based on vertical projection of the grid node coordinates on the triangular irregular network (TIN). This could result in an inappropriately high elevation assignment. Consider the case of an on-ramp to a highway that occupies only a portion of the grid system but happens to be located at the grid center. The grid element elevation would be based on the highway on-ramp elevation instead of the floodplain surface.

By selecting the standard deviation method for the filter system, all the DTM points within the interpolation radius are used in the computation of the standard deviation. Those DTM points that exceed the standard deviation are discarded and the remaining points are averaged to compute the grid element elevation. It is also possible that some DTM points may represent the channel bed elevation if the river is dry. Grid elements along the river can be assigned elevations that are lower than desired. Again by using the Low Elevation filter system, the lowest elevation DTM points can be eliminated from the interpolation of the floodplain elevation. These filtering schemes ensure that the floodplain surface is given preference when assigning floodplain elevations. Once the grid element elevations have been assigned, you can double-click on a grid element to observe its attributes. A dialog box is then displayed so that you can make any necessary changes:

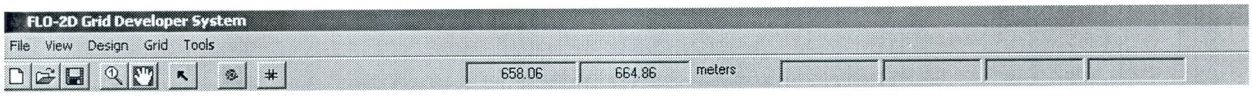


You may enter a new elevation or Manning's n-value for any individual grid element.

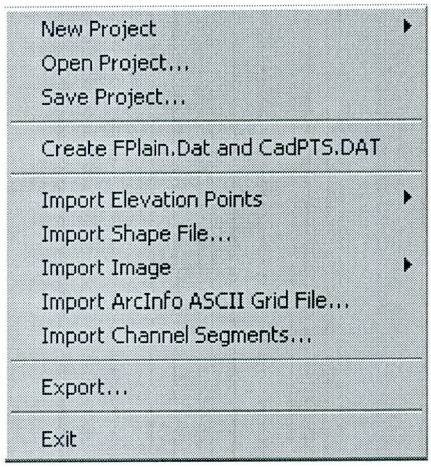
3.3.13 Detailed Description of the GDS Menus and Commands

A description of the main menu and the detailed description of each command are presented in the following sections. The command names and menu items are self-explanatory.

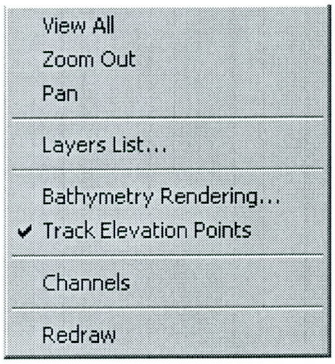
Main Menu



File Menu Commands:

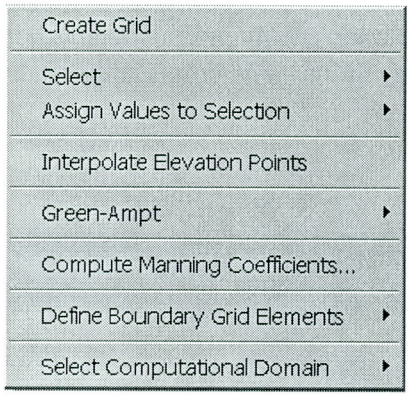


View Menu Commands:

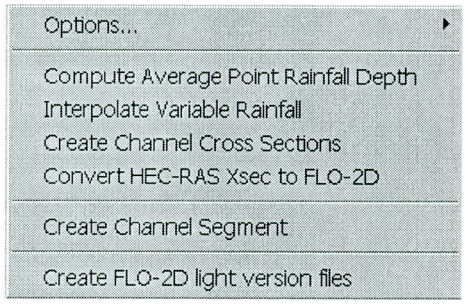


Design Menu Command:

Grid Menu Commands:

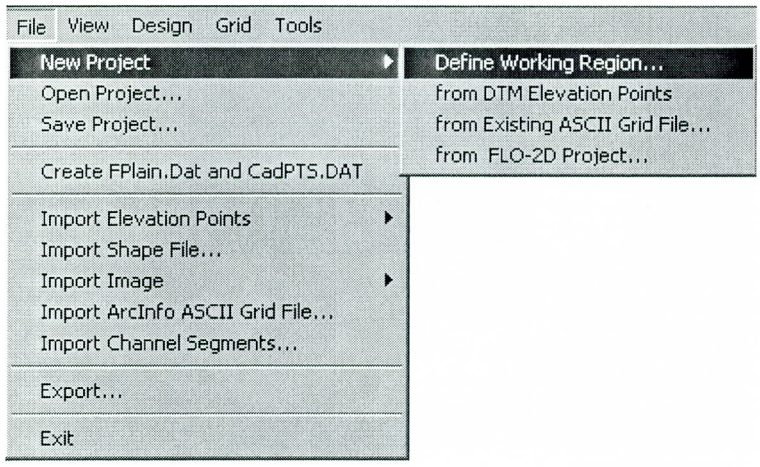


Tools Menu Commands:

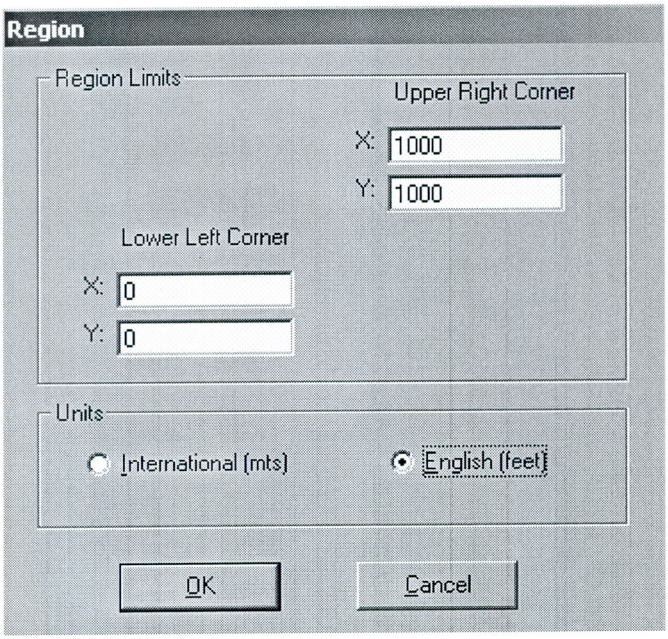


New Project/Define Working Region Command (File Menu)

This command creates a new project. You can only have one project open at a time. If there is a project open when this command is selected, GDS will prompt you to save the current project. You will then be prompted for the coordinates that define the working region of the new project.



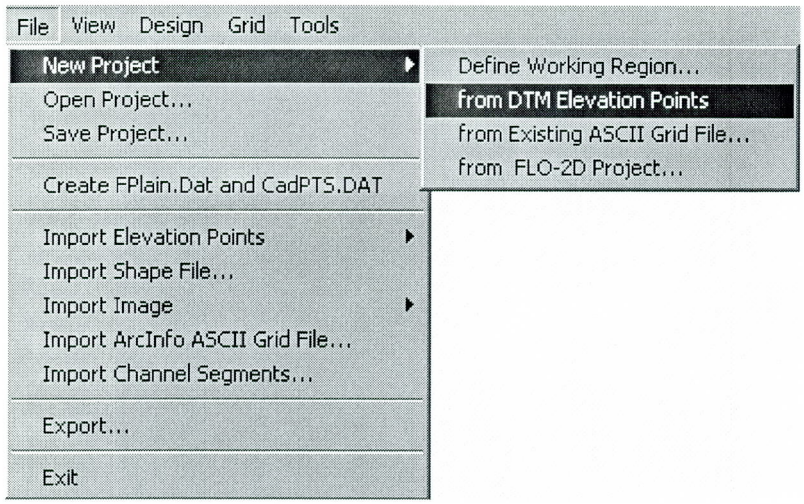
Region Limits Dialog Box:



Dialog Box Options:

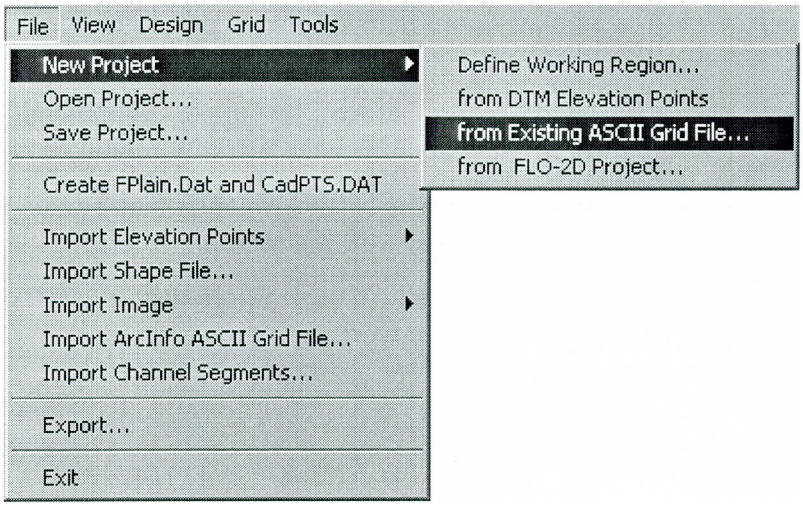
Option	**Description**
X (Upper Right Corner)	The X coordinate of the upper right hand corner of the rectangle that defines the working region.
Y (Upper Right Corner)	The Y coordinate of the upper right hand corner of the rectangle that defines the working region.
X (Lower Left Corner)	The X coordinate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

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



A new project is created by importing the DTM elevation points from an existing file. The working region is automatically scaled from the minimum and maximum point coordinates. You can only have one project open at a time. If there is a project already open when you select this command, GDS prompts you to save the current project. To import the DTM (*.PTS) file click on this command in the *File* menu and chose the correct filename in your project subdirectory.

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



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

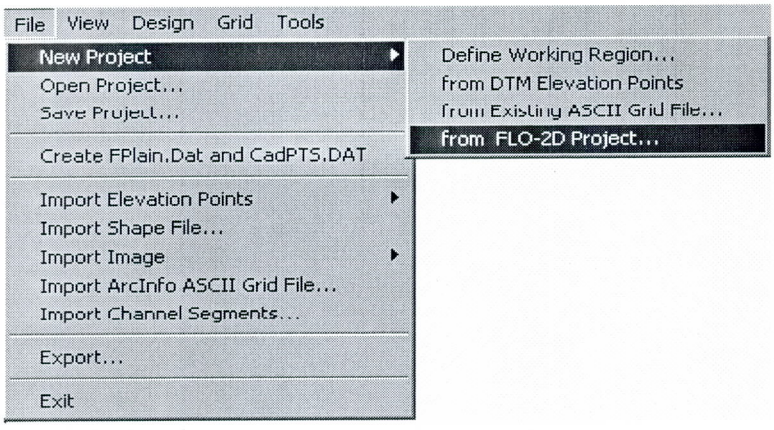
ncols /* Number of columns in the grid */
nrows /* Number of rows in the grid */
xllcorner x /* Lower left x coordinate of grid */
yllcorner y /* Lower left x coordinate of grid */
cellsize size /* Grid cell size */
NODATRA_value NODATA /* value of an empty grid cell */
z11 z12 z13 ... z1ncols /* values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

Rows are read from north to south. For example:

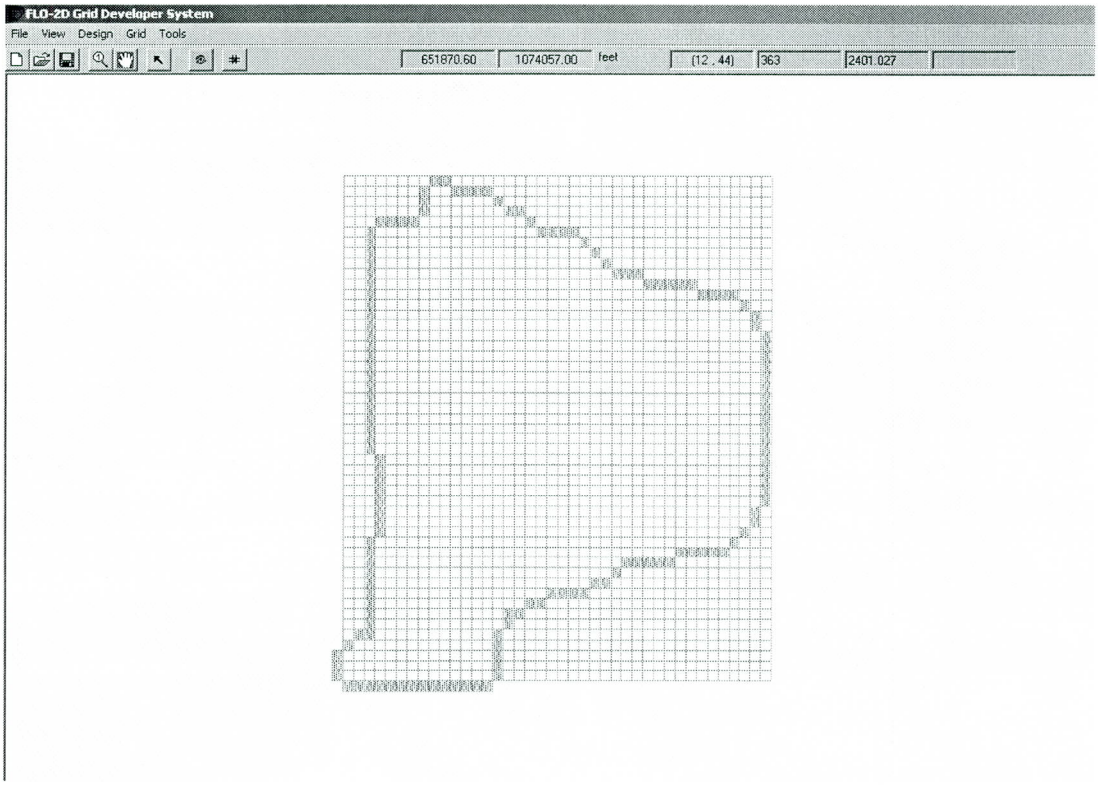
ncols 388
nrows 461
xllcorner 674070.85270015
yllcorner 1000118.1562353
cellsize 100
NODATA_value -9999
2477.259 2480.868 2486.877 2486.877 2487.308 2490.641 2493.438 2493.438
2493.438. . .

The project area is automatically scaled from the minimum and maximum point coordinates. To import the ArcInfo ASCII Grid File (*.ASC) file click on this command in the *File* menu and choose the correct filename in your project subdirectory. Please remember that you can only have one project open at a time. If there is a project open when you select this command, GDS prompts you to save the current project.

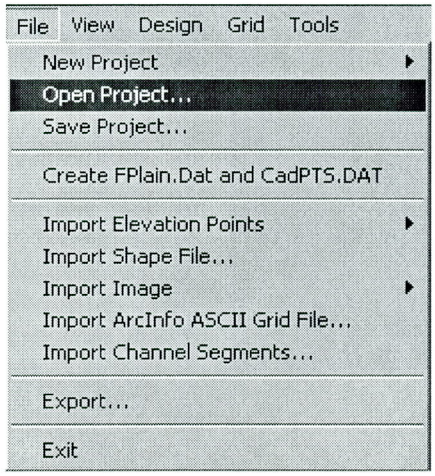
New Project/from FLO-2D Project...Command (File Menu)



A new project grid system can be created by importing an existing FLO-2D project. You can only have one project open at a time. If there is a project open when you select this command, GDS prompts you to save the current project. The working region and the FLO-2D grid are automatically scaled from the minimum and maximum coordinate points. To use this option, click on this command in the *File* menu and chose the FPLAIN.DAT file in your project subdirectory. The CONT.DAT data file must exist in the subdirectory. The resulting grid will look like this:

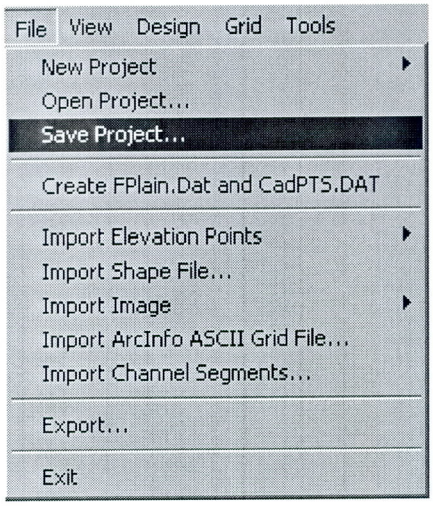


Open Project... Command (File Menu)



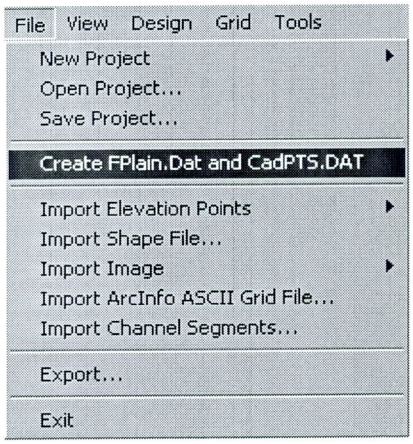
Use the *Open Project* command to open an existing project. You can only have one project open at a time. GDS will prompt you to save the current project before opening a new one. A dialog box will help you find the directory and the project file. All GDS project files have a *.TOP extension.

Save Project... Command (File Menu)



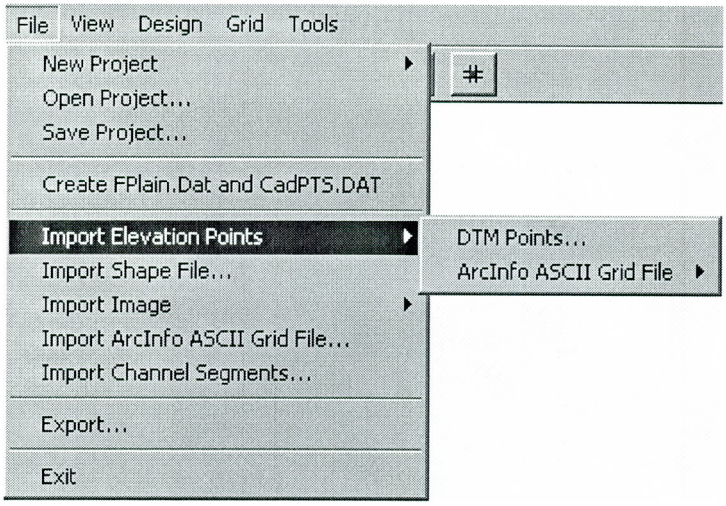
Use this command to save the active project. The system will display a file dialog so that a directory and file name can be selected. The project file name must have a *.TOP extension.

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



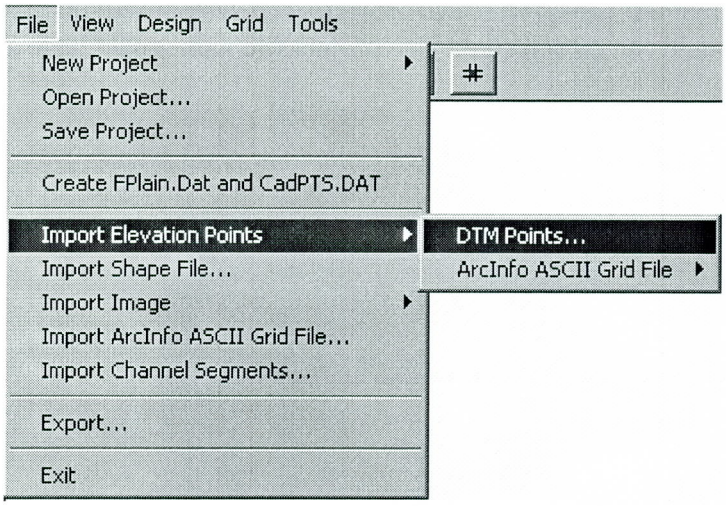
This command creates FPLAIN.DAT and CADPTS.DAT files required by FLO-2D.

Import Elevation Points Command (File Menu)



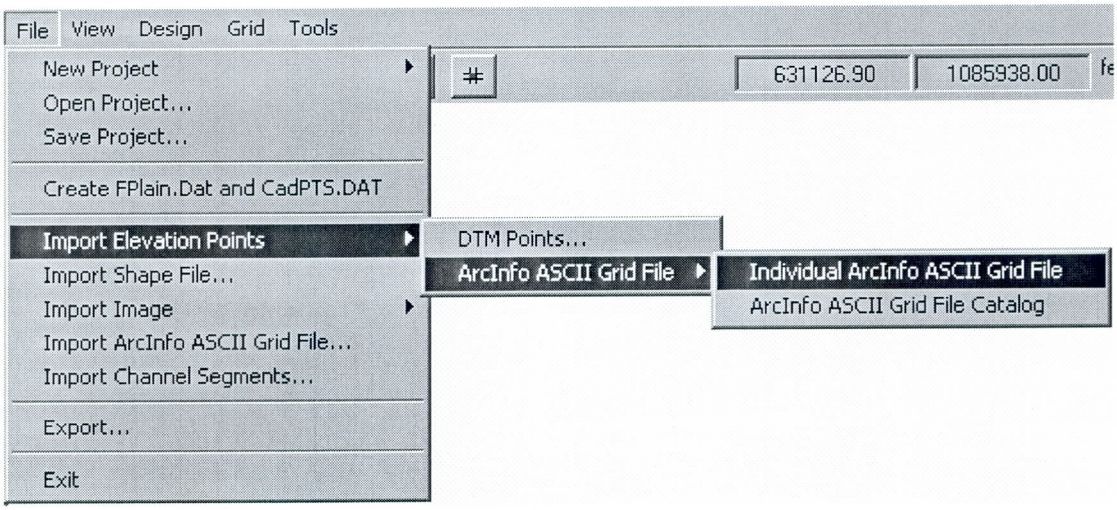
This command will import an existing terrain elevation file. It contains a submenu with two other commands to import either digital terrain elevation points (DTM Points Command) or an ArcInfo ASCII Grid File.

DTM Points...Command (File Menu):



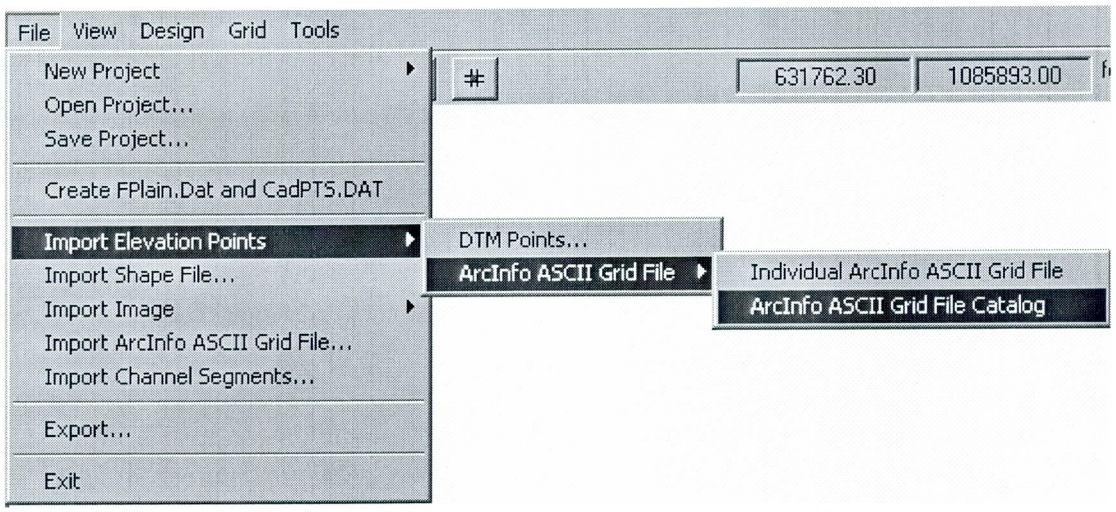
This command imports DTM elevation points from an existing file. You may import several data files. Any new points are added to the existing data points. You may also mix or combine DTM points with regularly located points coming from ArcInfo ASCII grid files.

Individual ArcInfo ASCII Grid File Command (File Menu) :

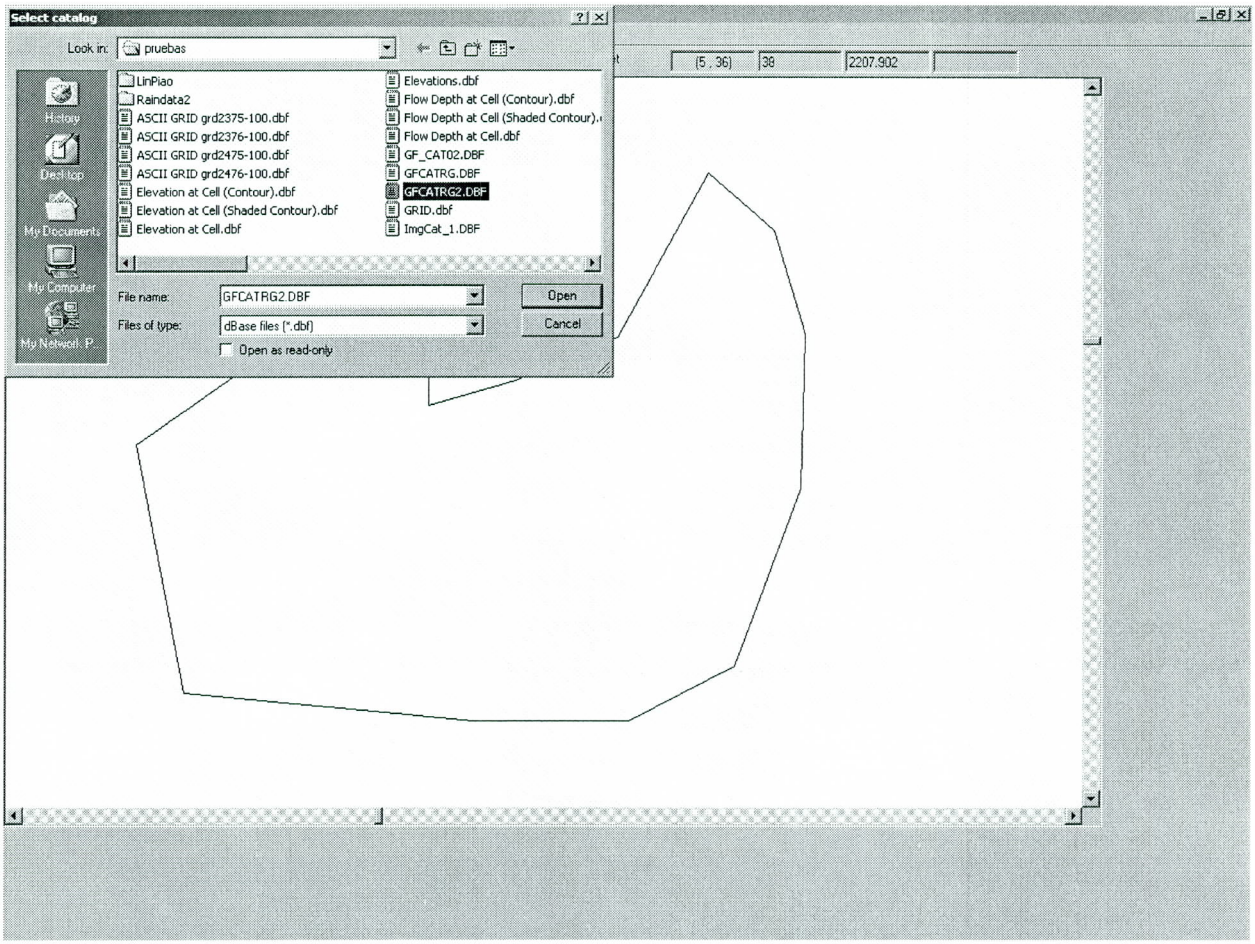


Use this command to import an individual ArcInfo ASCII grid files. You may import several grid files. Any new points are added to the existing data. You may also mix or combine this ArcInfo ASCII data with DTM points.

ArcInfo ASCII Grid File Catalog Command (File Menu):



This command allows you to import several ArcInfo ASCII grid files stored in any subdirectory. First draw a polygon on the working region, then select a catalog file.

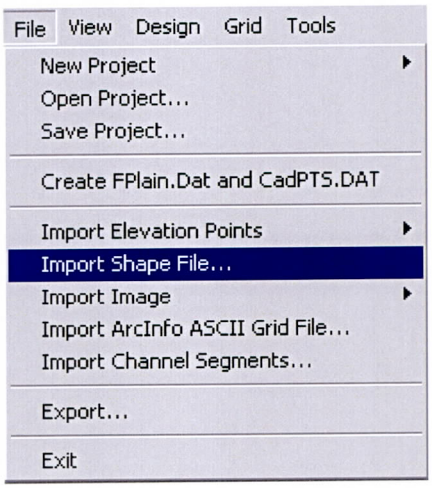


The catalog file may be in DBASE or ASCII format and has the following format:

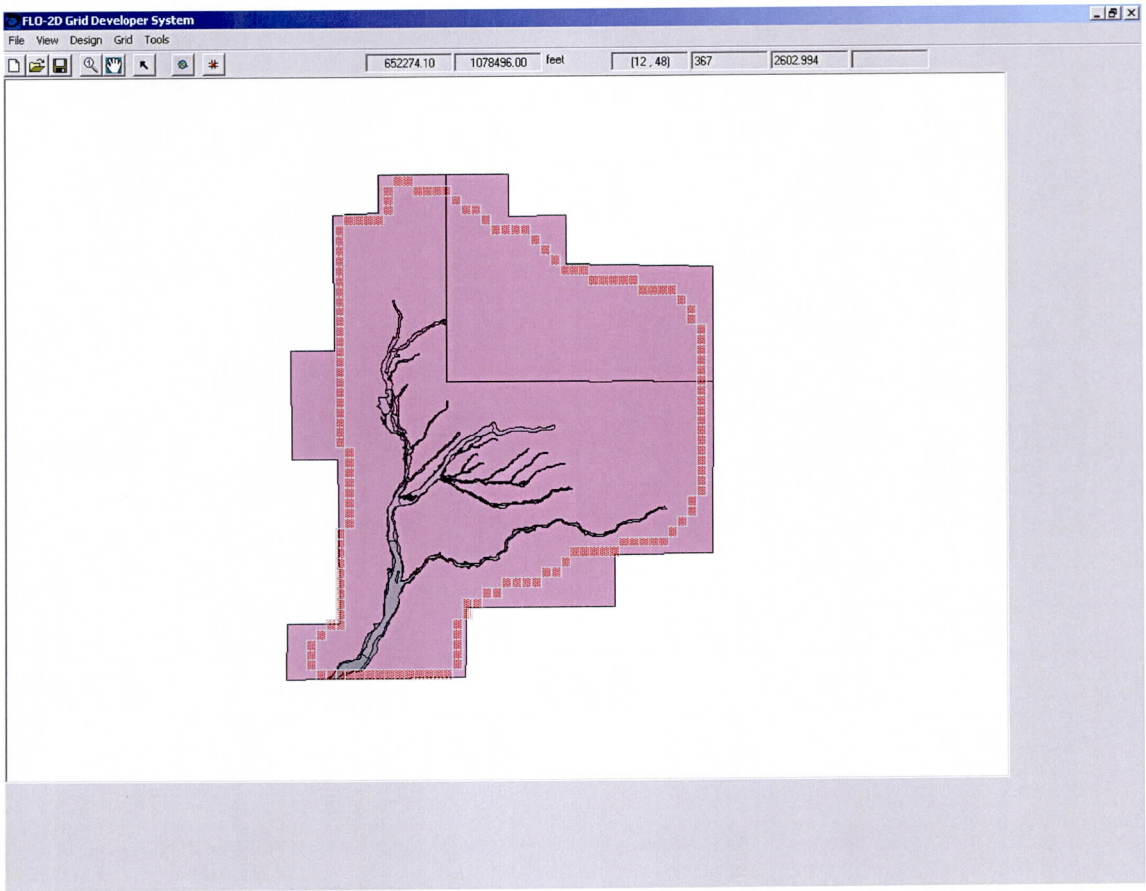
ASCII_GR	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\grd2375-100.asc	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\grd2376-100.asc	500000	1000000	900000	1500000
\\Agua\IMF\Publico IMF\grd2476-100.asc	510000	1000000	740000	1510000

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

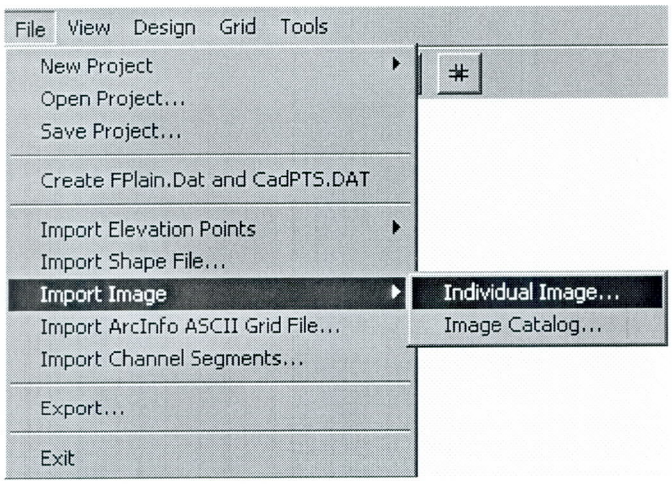
Import Shape File Command (File Menu)



This command will import ESRI shape files. The shape file data may represent soil properties, land use or Manning roughness coefficients. After the file has been read, the shape file is displayed on the screen. The following example is a shape file of Manning's n-values from the Flood Control District of Maricopa County, Phoenix, Arizona.



Import Image/Individual Image Command (File Menu)



Use this command to import individual images like aerial photos. You may import several images in the following formats:

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

To correctly locate the image or photo, it must be accompanied by a world file that contains its geo-reference data. This file has an extension depending on the file type.

The world file associated with an image is named by following the conventions in the table below. For example, if you have an image that's stored in a file named *myimage.bmp*, then the world file associated with it must be named *myimage.bmpw* or *myimage.bpw*

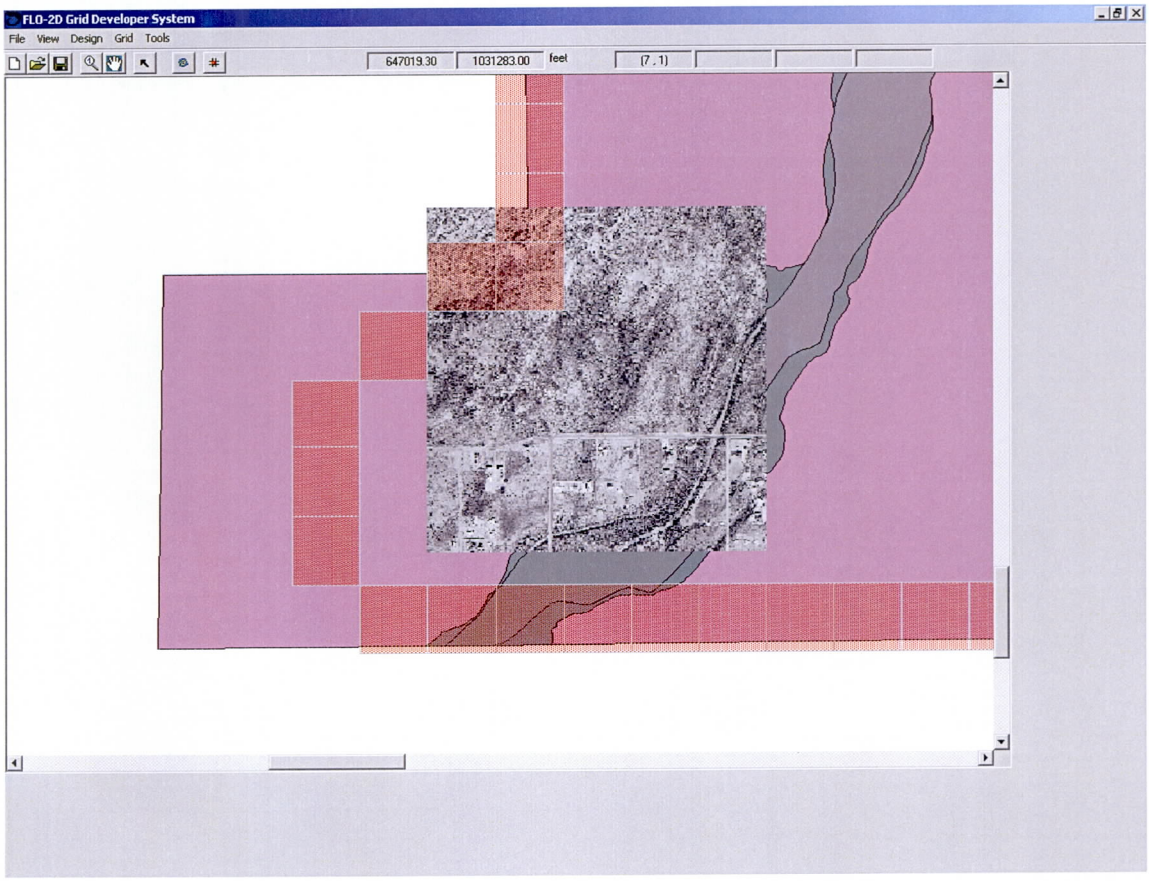
File Extension	World File Extension
bmp	bmpw or bpw
jpg; jpeg	jpgw or jgw
tif; tff; tiff	Tfw
gis	Gsw
lan	lnw
bil	blw
bip	bpw
bsq	bqw
sid	sdw
sun	snw
rs; ras	Rsw
rlc	rcw

The world file has the following general format:

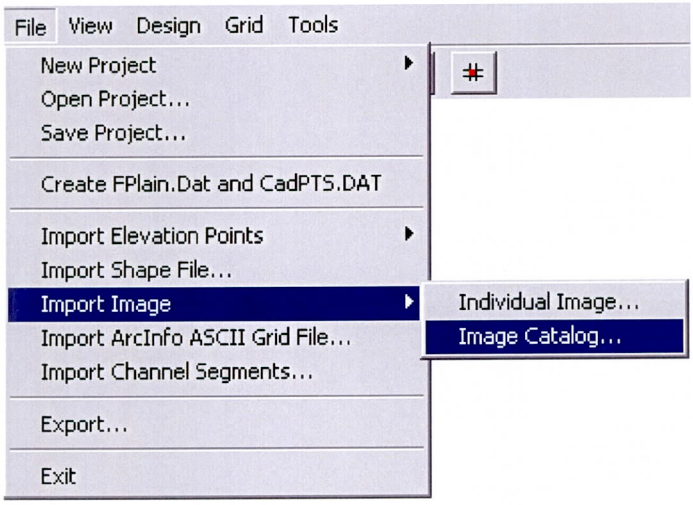
Line 1: Dimension of a pixel in map units in the x-direction.
Lines 2 and 3: Rotation terms. Not used in this release.
Line 4: This value is always negative because the image space is top-down whereas the map space is bottom-up.
Line 5: Translation term; x-Origin (x-coordinate of the center of the upper left pixel)
Line 6: Translation term; y-Origin (y-coordinate of the center of the upper left pixel)

Example:
20
0
0
-20
637510
1032490

An imported image is illustrated in the following figure:



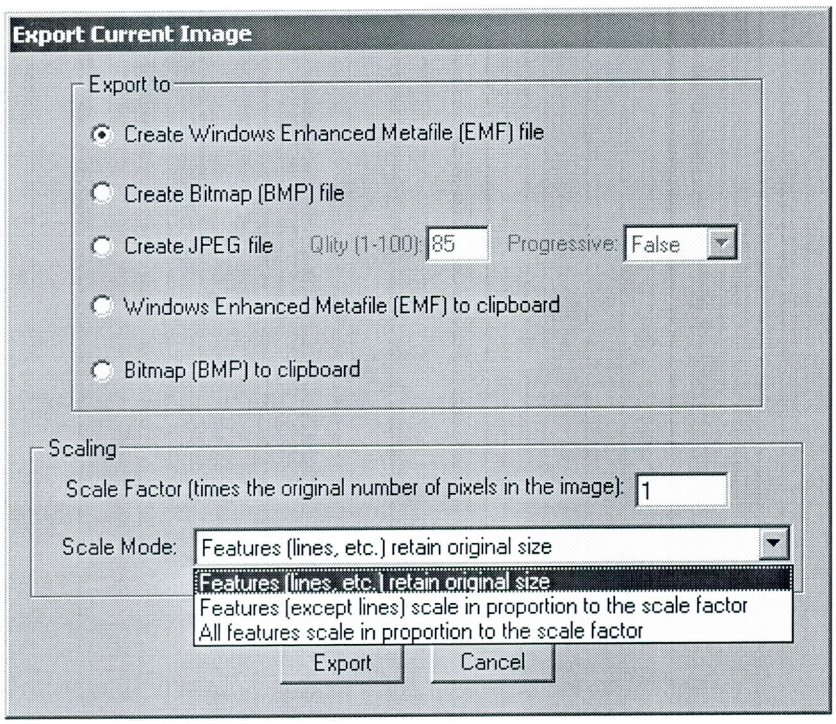
Import Image/Image Catalog Command (File Menu)



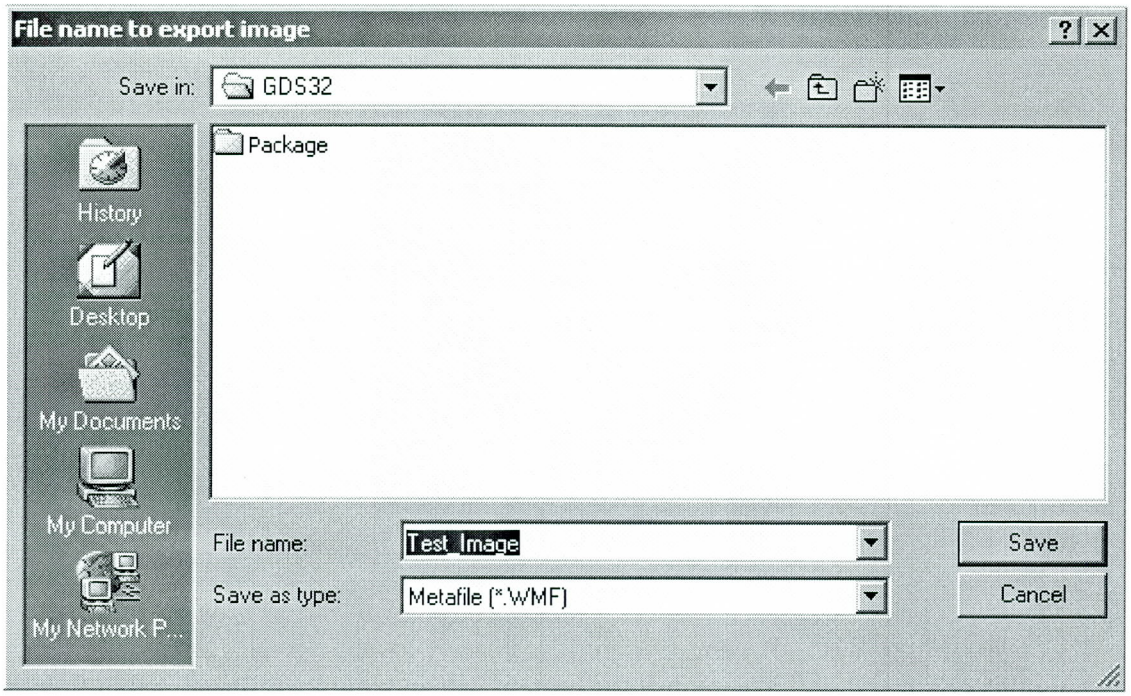
This command allows importing of several image files stored in any subdirectory. First

Export...Command (File Menu)

Using this command you may export the current screen view to different image format. When you click on the Export Command the following dialog box appears:



In this dialog box you may choose image export formats like Bitmap, JPEG, Windows Metafile, etc. You may also change the scale factor of the image to obtain better printing quality. When you click on the Export button, input the image file name and select the directory to save the image as shown in the following file dialog box:

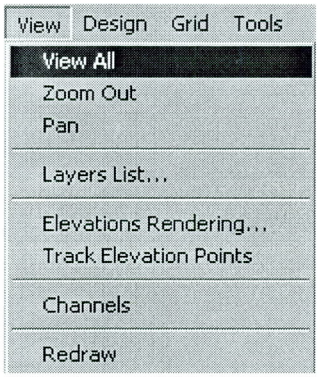


Exit Command (File Menu)

This command ends your GDS session. You can also use the *Close* command on the application Control menu. GDS will prompt you to save the current project.

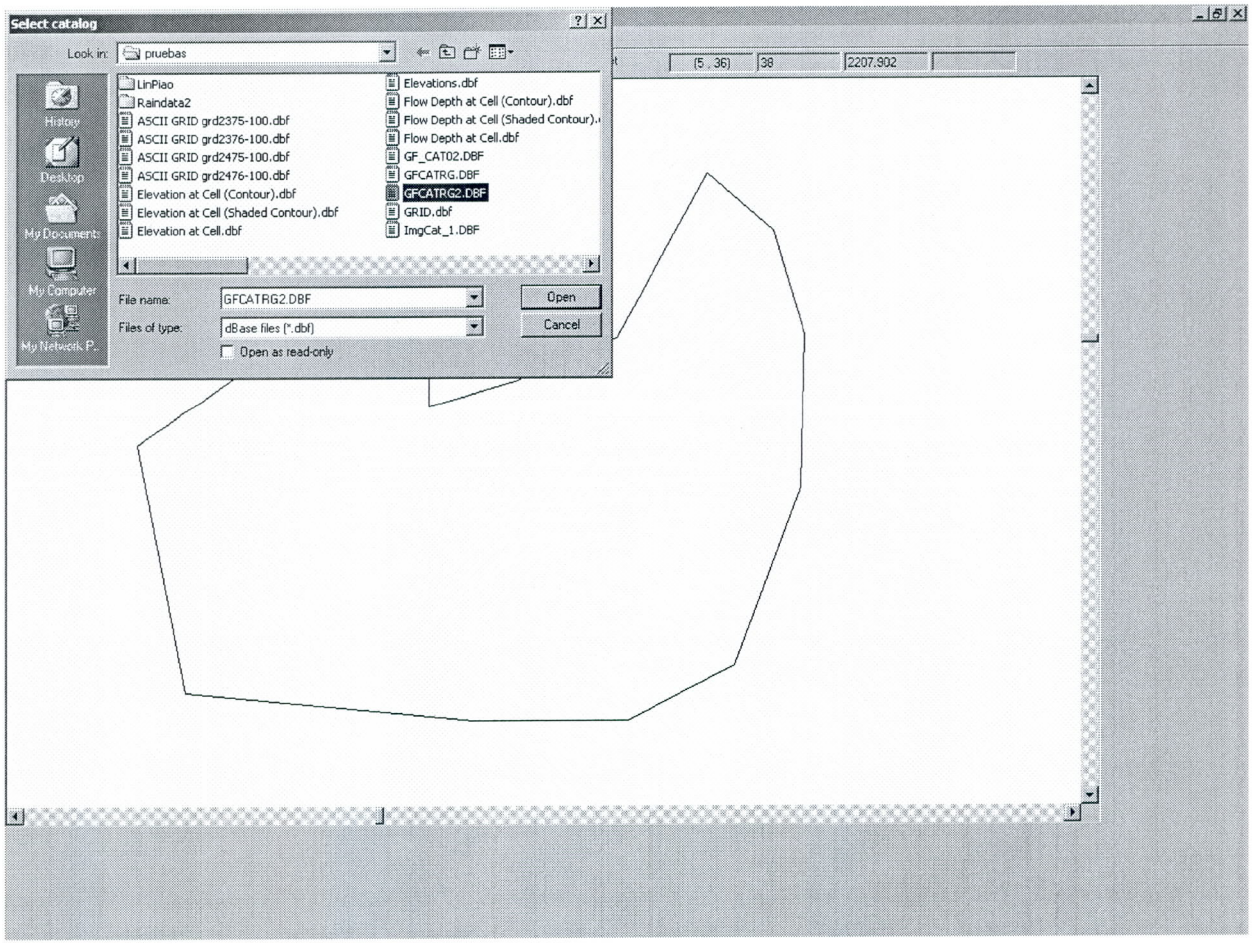
Shortcuts: Mouse: double-click the application Control-menu close button. Keys: Alt-F4

View All Command (View Menu)



Use the *View All* command or click on the *View All* toolbar icon to return the working region to its original full size. To Zoom-in (increase the magnification), just click in the working region and drag the mouse to outline the area you want to magnify.

draw a polygon on the working region and then select an image catalog file.



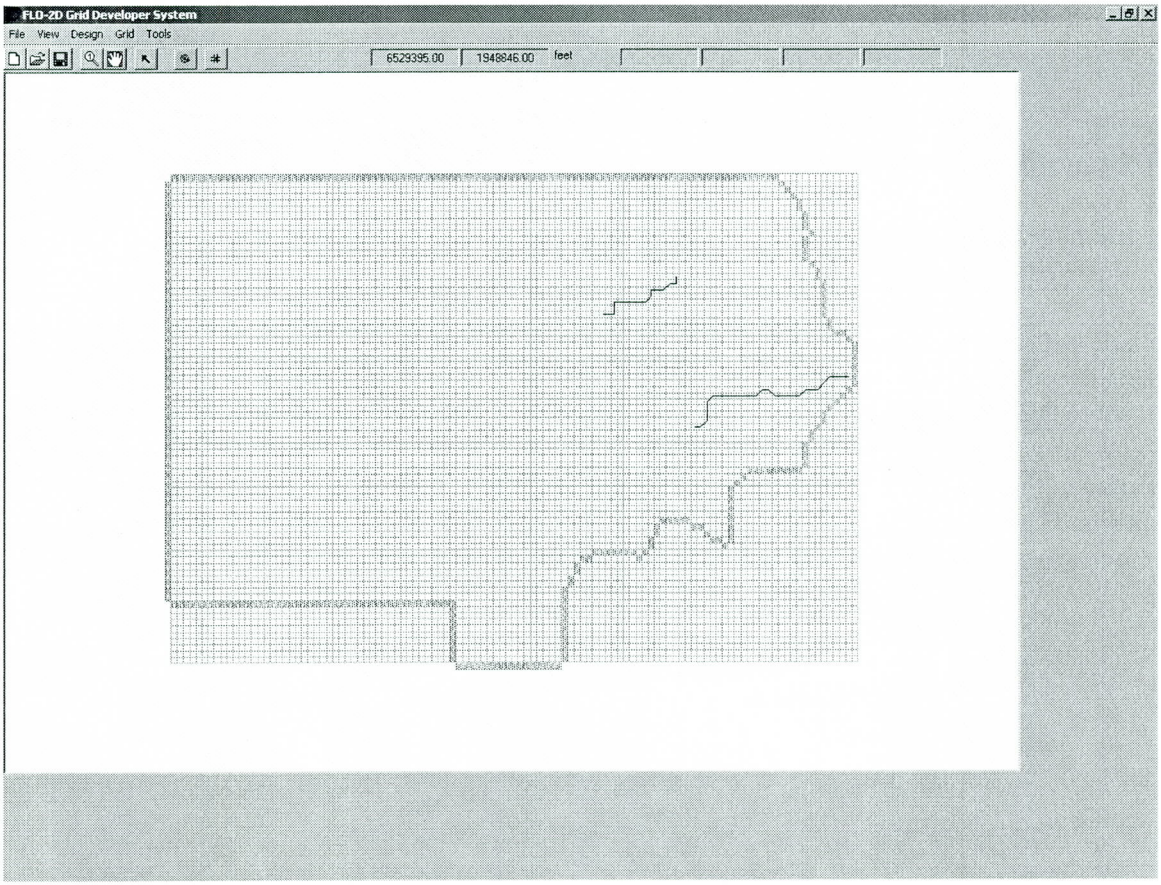
The catalog file may be in DBASE or ASCII format and has the following format:

IMAGE	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\ 6401030-5.TIF	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\ 6401035-5.TIF	660000	1300000	770000	1500000
\\Agua\IMF\Publico IMF\ 6401055-5.TIF	630000	1300000	750000	1510000

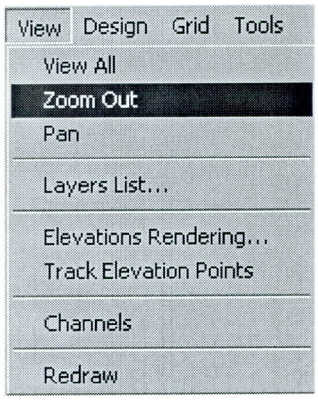
The first column is the file name including its path and the following four columns are the image coordinate limits. The GDS will find all images from the catalog that are contained within or intersected by the user defined polygon and will retrieve the corresponding images.

Import Channel Segments Command (File Menu)

This command retrieves all channel data from the FLO-2D CHAN.DAT file that will be used to extract additional channel cross sections. The GDS will plot the channel segment locations over the grid system.

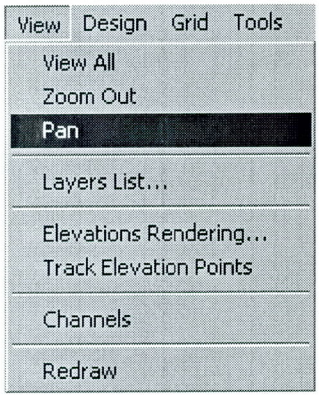


Zoom Out Command (View Menu)



Use the *Zoom Out* command to return the working region to its previous zoom extent.

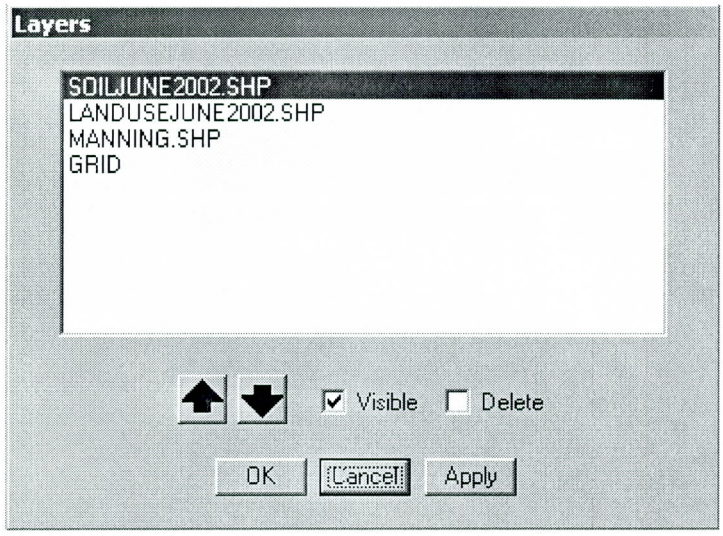
Pan Command (View Menu)



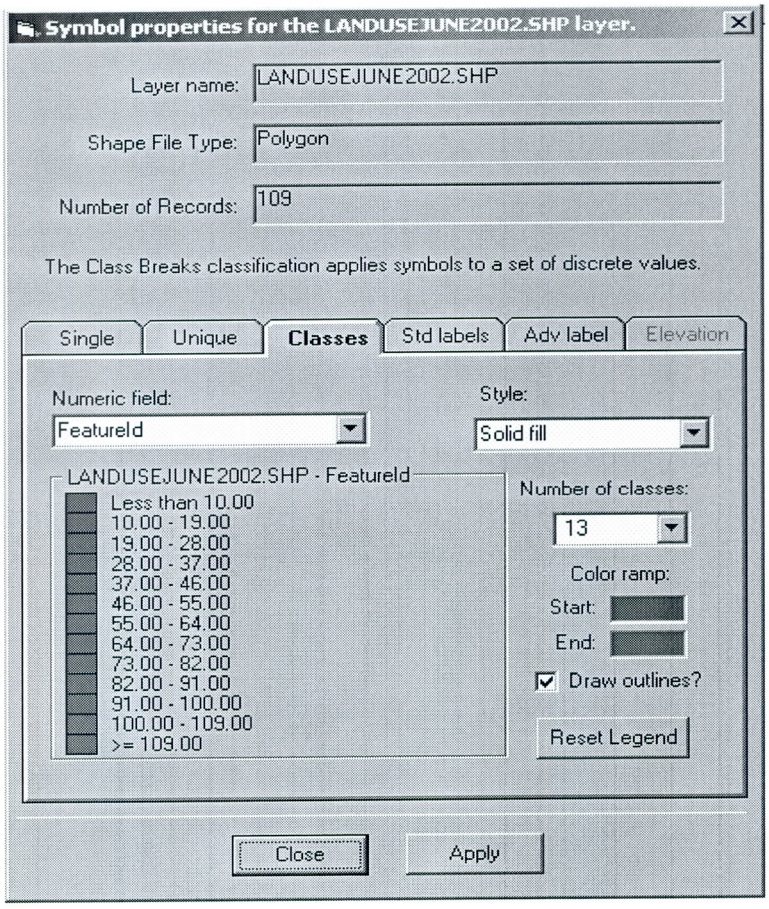
Use the *Pan* command to move around within the working region view. Click and drag the mouse to move around within the view. You can also use the *Pan* toolbar icon for the same purpose. Use the *View All* command (or toolbar icon) to return to full view of the working region.

Layers List Command (View Menu)

This command opens the Layer dialog box.

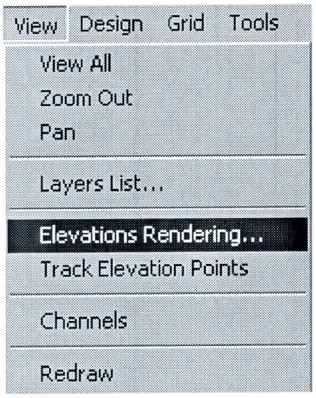


Layers may be visible or invisible. You can change the visible status of a layer by checking the *Visible* check box. In the example above there are four active layers. You may use the and icons to highlight and move between the various layers. Click on the *Apply* button to accept the changes. You may also delete any layer by checking the *Delete* check box and then clicking the *Apply* button. To enter the Layer Management Dialog box double click on an active layer. The following dialog appears:

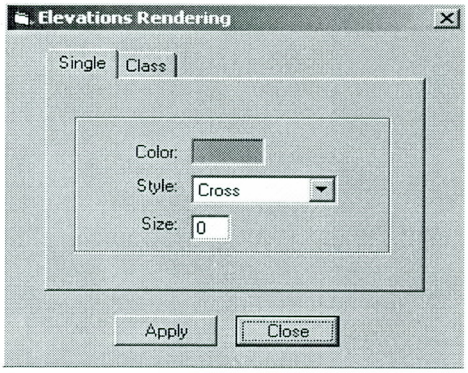


In this dialog box you may change layer colors, transparency, number of classes or divisions, add point captions, font styles, etc. Click on the *Apply* button and then the *Close* button to accept the changes.

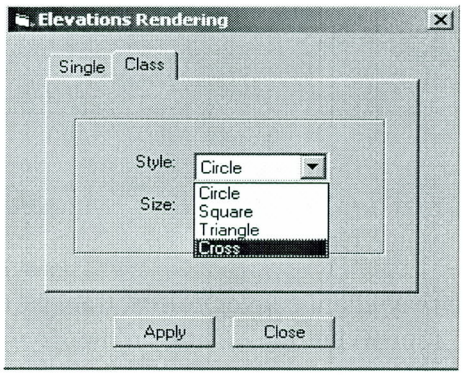
Elevations Rendering Command (View Menu)



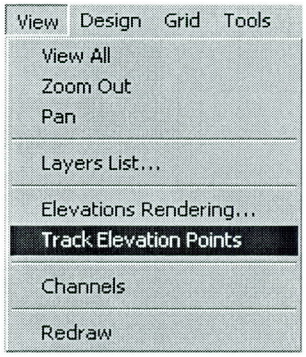
This command selects the elevation point display including color, style and size.



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

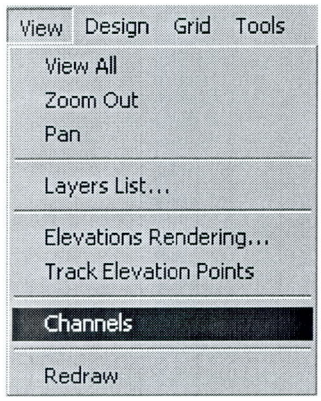


Track Elevation Points Command (View Menu)



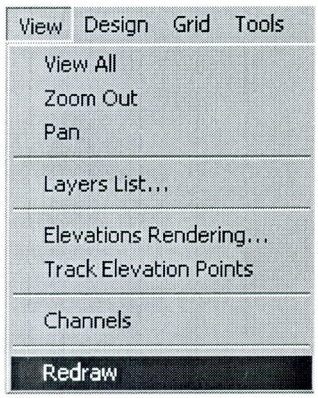
This command queries individual terrain elevation points and displays the data for the selected point. The mouse cursor changes to the inquiry mode to show that this option is active.

Channels Command (View Menu)



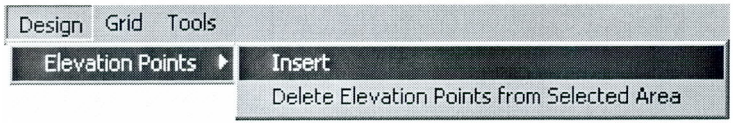
This command toggles on and off the display of channel segments from the CHAN.DAT file if the data file has been created. When the command is checked, the working region shows channel segments.

Redraw Command (View Menu)



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

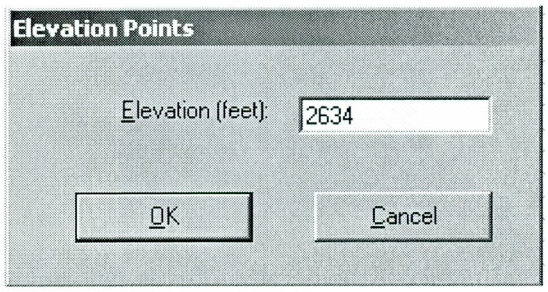
Elevation Points/Insert Command (Design Menu)



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

1. Select the *Elevation Points/Insert* command (Design menu) or the toolbar icon and click the left mouse button.

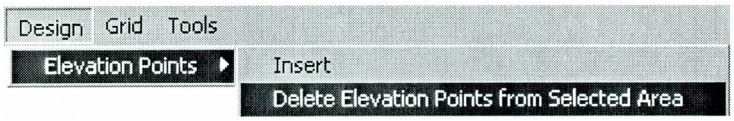
2. A dialog box appears for the elevation entry in feet or meters. Click on the 'OK' button to accept the value.



3. Click a point on the working region where you want to assign this elevation data.

4. Repeat step 3 as many times as needed.

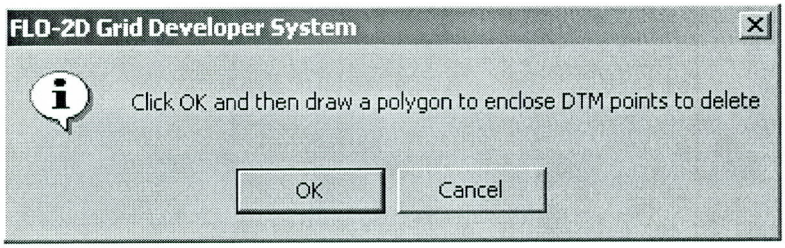
Elevation Points/Delete Elevation Points from Selected Area Command (Design Menu)



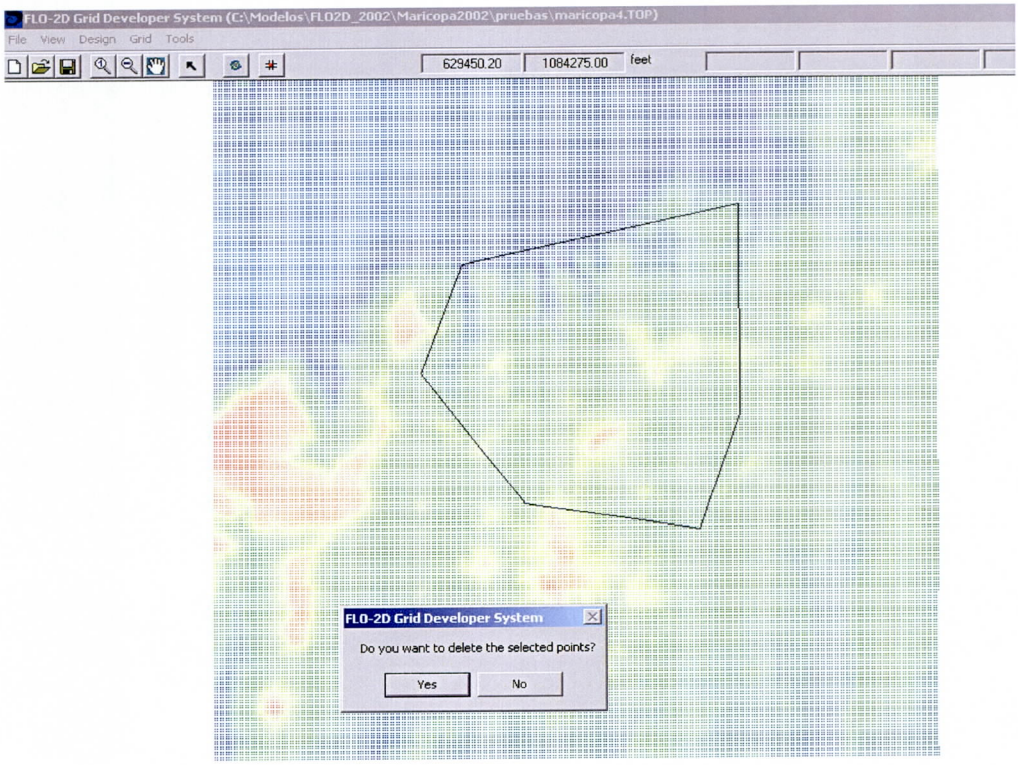
This command deletes elevation data points within the working region. To use this tool:

1. Select the *Elevation Points/ Delete Elevation Points from Selected Area* command (Design menu) and click the left mouse button.

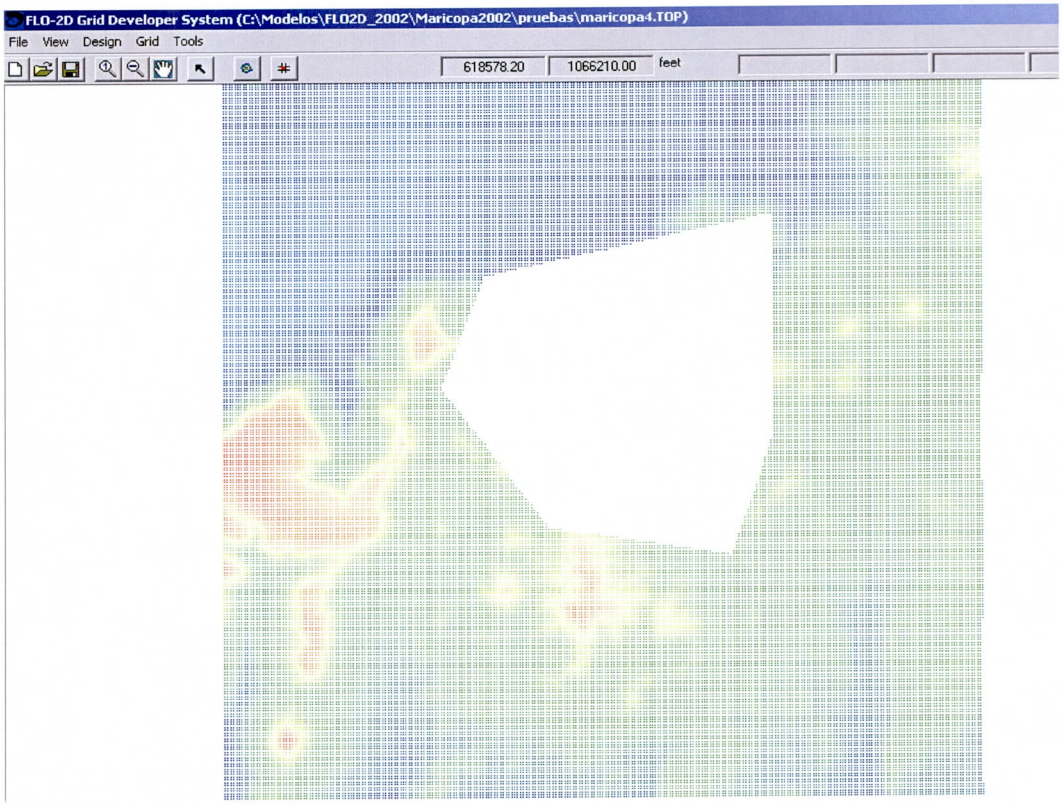
2. The following dialog box appears.



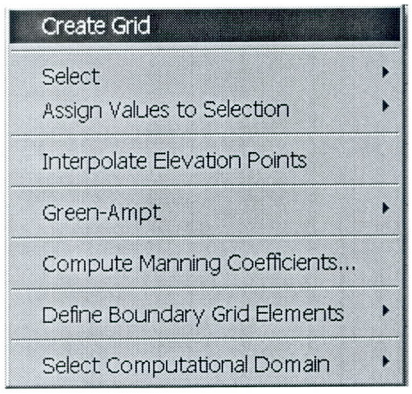
3. Draw a polygon ending with a double click to close the polygon.



4. Click OK to delete the points inside the polygon as shown in the following figure.



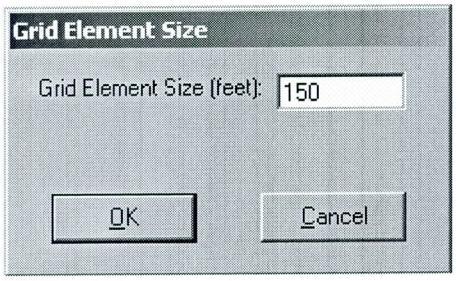
Create Grid Command (Grid Menu)



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

1. Select the *Create Grid* command (Grid menu).

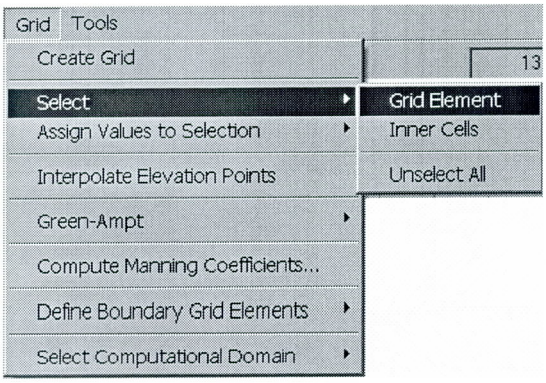
2. A dialog box appears requesting the square grid element size or side length (meters or feet):



3. Select OK to accept the value.

4. The GDS system will automatically overlay a grid template that is centered in the working region. The grid elements will be centered on each node.

Select/Grid Element Command (Grid Menu)

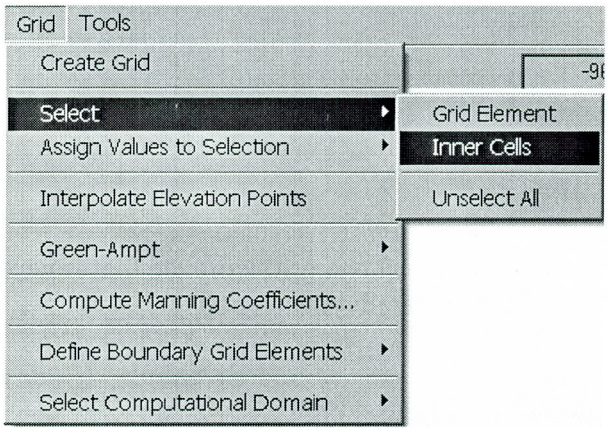


Use this command to select one or more grid elements. The selected elements are shown as cross-hatched with diagonal lines. With the *Assign Values to Selection* command, you can assign attribute values to the selected grid elements. To use the *Select / Grid element* command:

1. Choose the *Select/Grid element* command (Grid menu).
2. The cursor changes to a cross.
3. Mouse click on the grid element you want to select.
4. Repeat step 3 for each selected element.

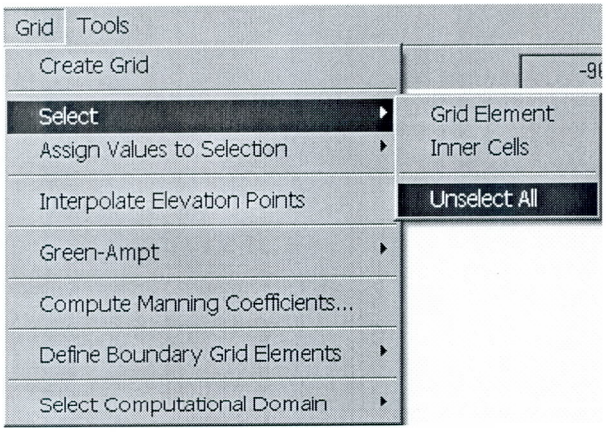
To unselect previously selected elements, simply click on them again. To select a group of elements, press the **Shift key** simultaneously with the left mouse button and drag the mouse pointer over the desired elements. To unselect a group of elements press the **Control key** and the left mouse button, then drag the mouse pointer over the elements you want to unselect. When you are dragging the mouse over the grid elements, they are painted to indicate your selection. After a grid element or group of elements is selected, you may use the *Assign Values to Selection* command to assign a Manning's roughness coefficient or an elevation value to the selected elements.

Select/Inner Cells Command (Grid Menu)



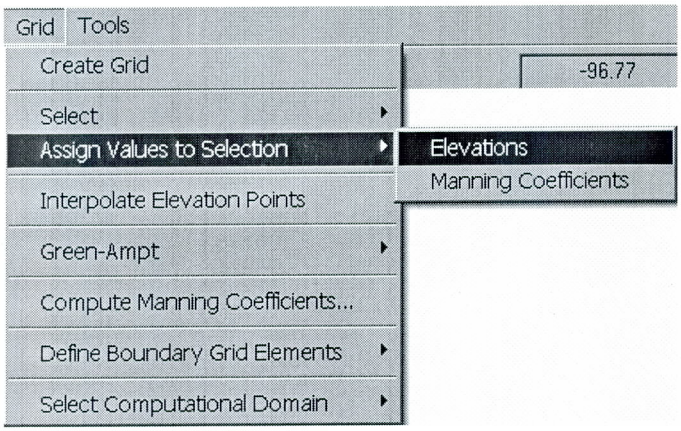
This command will select all the grid elements within the computational domain and hatch them with diagonal lines. Use the *Assign Values to Selection* command to assign a Manning's n-value or an elevation to the elements.

Select/Unselect All Command (Grid Menu)

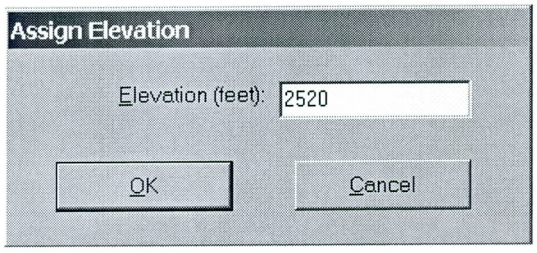


This command will unselect all the elements previously selected with the *Select* command.

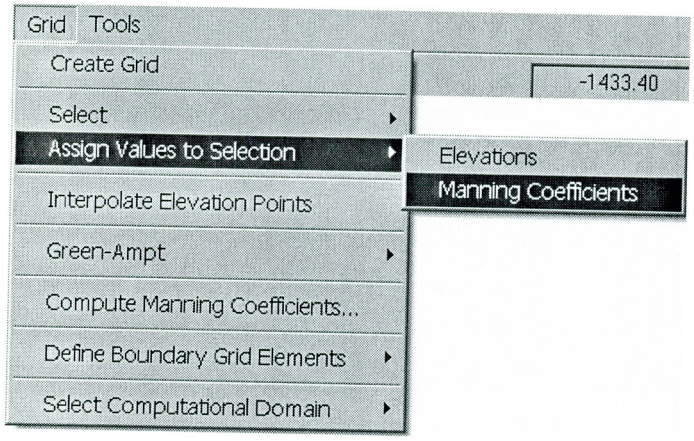
Assign Values to Selection/Elevations Command (Grid Menu)



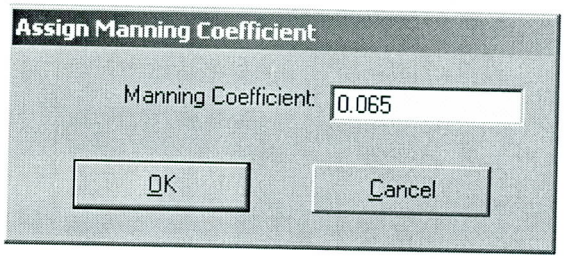
With this command you can assign an elevation to elements that have been selected using the *Select* command. Upon selection, a dialog box appears prompting you to enter an elevation. Press 'OK' to assign the elevation to all the selected elements:



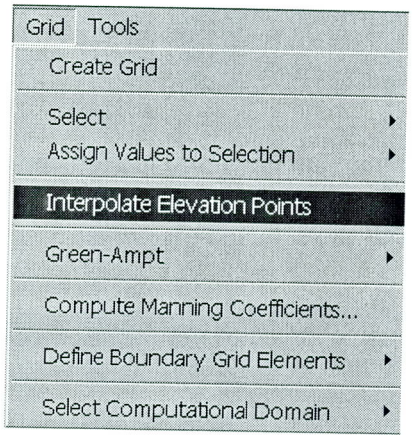
Assign Values to Selection/Manning's Coefficient Command (Grid Menu)



A Manning's roughness coefficient (n-value) is assigned to the grid elements previously selected with the *Select* command. A dialog box appears prompting you to enter the n-value:

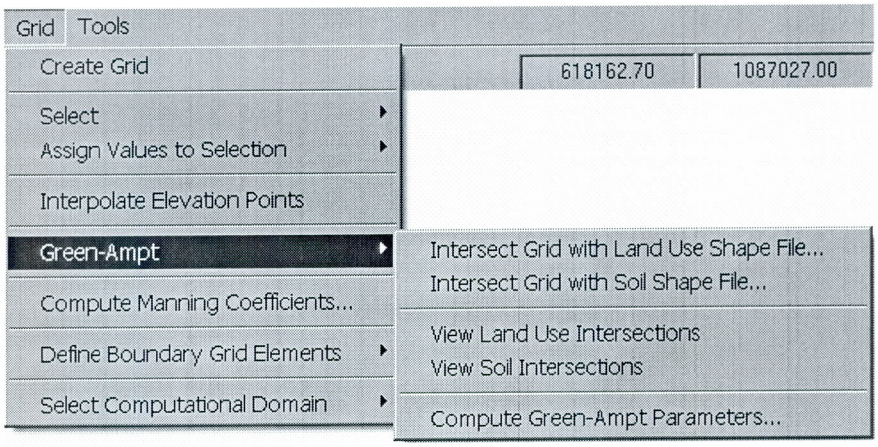


Interpolate Elevation Points Command (Grid Menu)



Use this command to interpolate and assign all the grid element elevations in the computational domain. The interpolated elevations are based on the imported DTM points or the elevation points that are added to the working region using the *Elevation Point / Insert* command. **The interpolation and assignment of elevations using this command must be completed before the FPLAIN.DAT and CADPTS.DAT data files can be created.** The interpolation options are selected in the *Options/Interpolation* command of the *Tools* menu.

Green-Ampt Command (Grid Menu)



This command is the starting point of the Green-Ampt parameter calculations. There are several steps that have to be completed in a prescribed sequence. The final result is the creation of the FLO-2D INFIL.DAT file containing the Green-Ampt infiltration parameters. To initiate this procedure, you have to load a land use shape file and a soil shape file. Soil and land use tables also must be available. In addition, the FLO-2D grid system data files must exist.

The general procedure is as follows:

1. Read soil shape file.
2. Read land use shape file.
3. Determine soil type polygon intersections with the FLO-2D grid elements.
4. Determine land use polygon intersections with the FLO-2D grid elements.
5. For each grid element compute hydraulic conductivity (*XKSAT)* average.

$$\overline{XKSAT} = ALOG\left(\frac{\sum A_i \log(XKSAT_i)}{A_{GE}}\right)$$

where $XKSAT_i$ is obtained from the soil table, A_i is subarea intercepted by the grid element from the 3rd column of the land use table and A_{GE} the grid element area.

6. For each grid element, compute wetting front capillary suction PSIF according to the following regressions as a function of *XKSAT* (Generated from Figure 4.3 of the Maricopa County Drainage Design Manual, Volume I).

XKSAT (in/hr)	**PSIF** (in)
$0.01 \le XKSAT \le 1.2$	$PSIF=EXP(0.9813-0.439*Ln(XKSAT)+0.0051(Ln(xksat))^2+0.0060(Ln(XKSAT))^3)$

7. For each grid element, compute volumetric soil moisture deficiency *(DTHETA)* according to the following table. The specific table used for DTHETA depends on the *saturation* field of the soil table (6th column).

Saturation = DRY

XKSAT (in/hr)	**DTHETA DRY**
$0.01 \le XKSAT \le 0.15$	DTHETA =EXP(-0.2394 + 0.3616 Ln(XKSAT))
$0.15 < XKSAT \le 0.25$	DTHETA =EXP(-1.4122 - 0.2614 Ln(XKSAT))
$0.25 < XKSAT \le 1.2$	DTHETA = 0.35

Saturation = NORMAL

XKSAT (in/hr)	**DTHETA NORMAL**
$0.01 \le XKSAT \le 0.02$	DTHETA = EXP(1.6094 + Ln(XKSAT))
$0.02 < XKSAT \le 0.04$	DTHETA = EXP(-0.0142 + 0.5850 Ln(XKSAT))
$0.04 < XKSAT \le 0.1$	DTHETA = 0.15
$0.1 < XKSAT \le 0.15$	DTHETA = EXP(1.0038 + 1.2599 Ln(XKSAT))
$0.15 < XKSAT \le 0.4$	DTHETA = 0.25
$0.4 < XKSAT \le 1.2$	DTHETA = EXP(-1.2342 + 0.1660 Ln(XKSAT))

Saturation = WET or SATURATED

DTHETA = 0 for all XKSAT

8. Adjust *XKSAT* (computed in step No. 1) as a function of the vegetation cover VC from the 5th field of the land use table when XSAT < 0.4 in/hr. This requires a computation of the ratio the hydraulic conductivity for the vegetative cover to the bare ground hydraulic conductivity (C_K):

$$C_K = \frac{VC_K - 10}{90} + 1$$

$$XKSATC = XKSAT \sum_k P_k C_k$$

Where P_k is the percentage of the area within the grid element corresponding to C_k. XKSATC for each grid element is written to the INFIL.DAT file.

9. For each grid element compute the initial abstraction *IABSTR*:

$$IABSTR = \left(\frac{\sum A_i (IA_i)}{A_{GE}} \right)$$

where IA_i is the initial abstraction in the subarea A_i intercepted by the element and is based on the 3rd column of the land use table. The intercepted subareas are computed using the land use shape file. *IABSTR* is added to the INFIL.DAT file for each element.

10. Compute effective impervious area (%) for each grid element *(RTIMP)*.

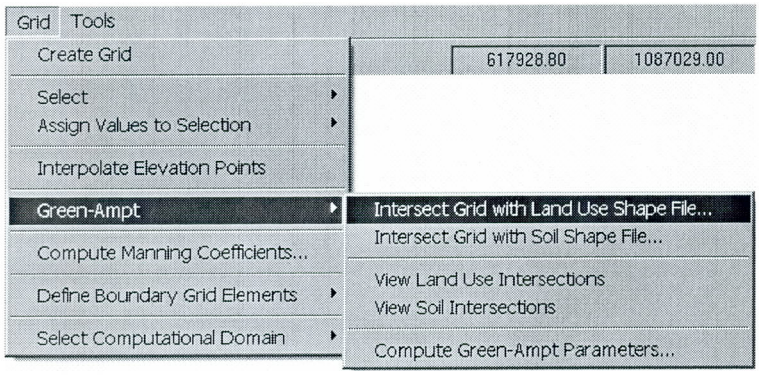
$$RTIMP_1 = \left(\frac{\sum A_i (RTIMPS * EFF)_i}{A_{GE}} \right)$$

where *Ai* is determined from the soil shape file, A_{GE} is the grid element area, effective impervious area *EFF* is obtained from the 5th column of the soil table and the percent rock outcrop *RIMPS* is obtained from the 4th column of the soil table.

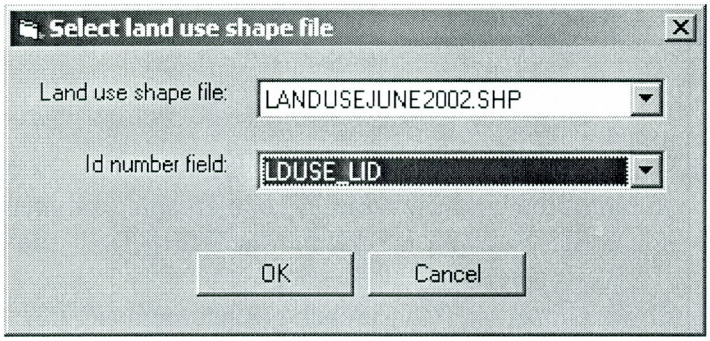
$$RTIMP = RTIMP_1 + \left(\frac{\sum A_i (RTIMPL)_i}{A_{GE}} \right)$$

where *Ai* is obtained from land use shape file, A_{GE} is the grid element area and *RTIMPL* is obtained from the 4th column of the land use table.

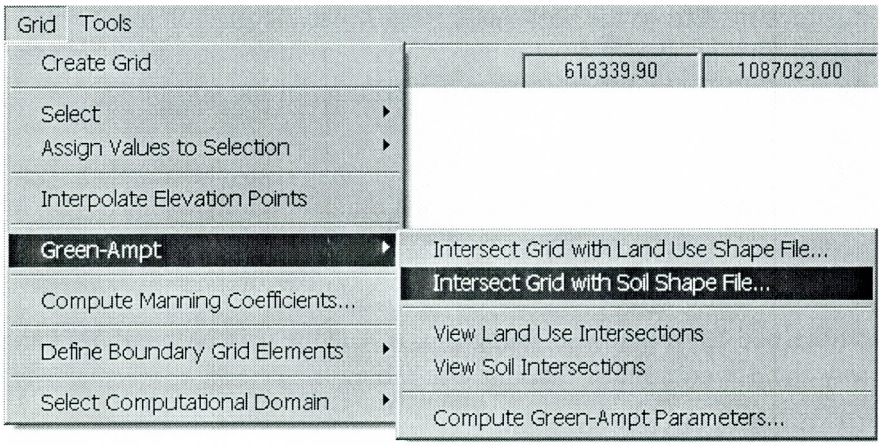
The steps to complete Green-Ampt parameter computation are as follows. Compute polygon intersections of the land use shape file with each grid element.



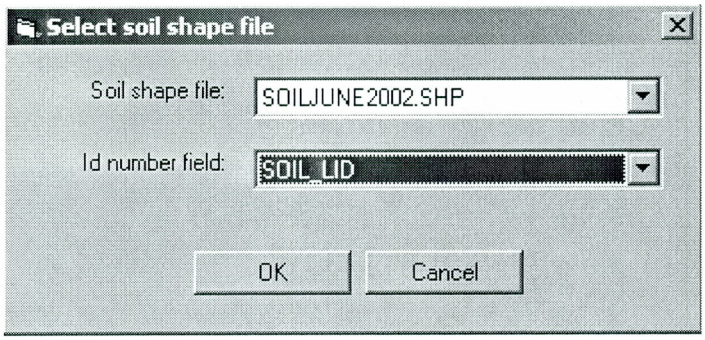
Specify the land use shape file and the ID number field as shown:



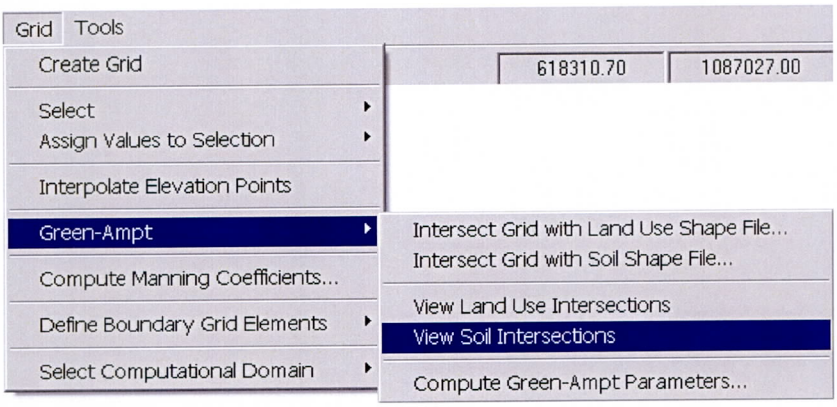
Click 'OK' to compute. This procedure may take several minutes depending on the number of grid elements and the number of land use polygons. Now compute the polygon intersections of the soil shape file with each grid element.



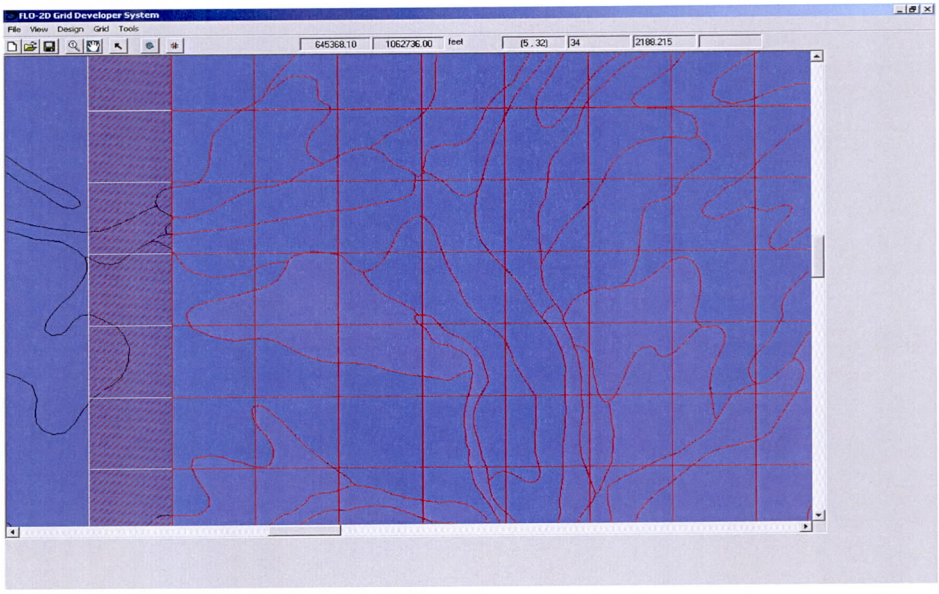
Next, it is necessary to specify the soil shape file and the ID number field:



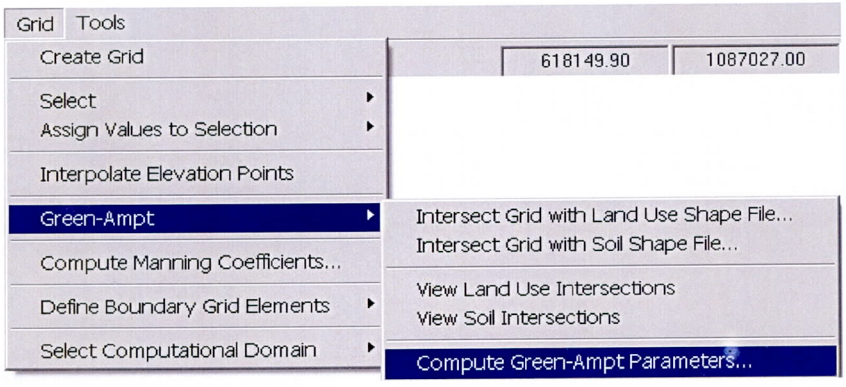
Click 'OK' to compute. This procedure may take several minutes depending on the number of grid elements and soil type polygons. The computed intersected polygons can be viewed.



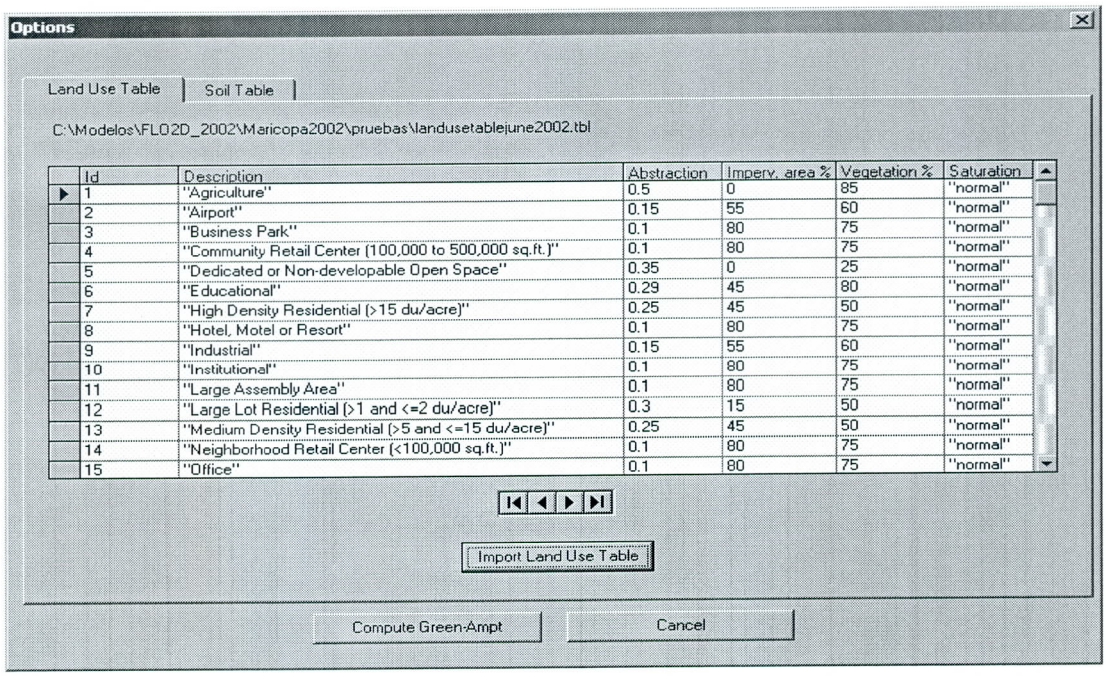
This is a zoom view showing soil polygons intersected with grid elements.



Now click on the Compute Green-Ampt Parameters.

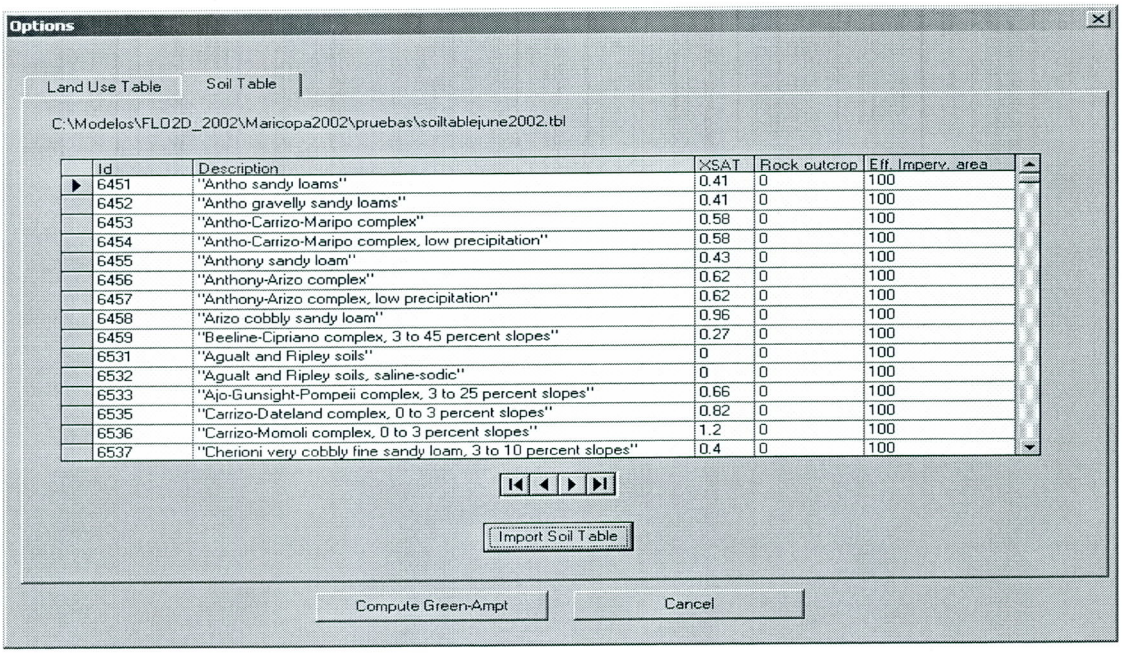


It is necessary to import the soil and land use tables. First click on Import Land Use Table button and select the land use table file to import. The data is displayed in the dialog box:



Id	Description	Abstraction	Imperv. area %	Vegetation %	Saturation
1	"Agriculture"	0.5	0	85	"normal"
2	"Airport"	0.15	55	60	"normal"
3	"Business Park"	0.1	80	75	"normal"
4	"Community Retail Center (100,000 to 500,000 sq.ft.)"	0.1	80	75	"normal"
5	"Dedicated or Non-developable Open Space"	0.35	0	25	"normal"
6	"Educational"	0.29	45	80	"normal"
7	"High Density Residential (>15 du/acre)"	0.25	45	50	"normal"
8	"Hotel, Motel or Resort"	0.1	80	75	"normal"
9	"Industrial"	0.15	55	60	"normal"
10	"Institutional"	0.1	80	75	"normal"
11	"Large Assembly Area"	0.1	80	75	"normal"
12	"Large Lot Residential (>1 and <=2 du/acre)"	0.3	15	50	"normal"
13	"Medium Density Residential (>5 and <=15 du/acre)"	0.25	45	50	"normal"
14	"Neighborhood Retail Center (<100,000 sq.ft.)"	0.1	80	75	"normal"
15	"Office"	0.1	80	75	"normal"

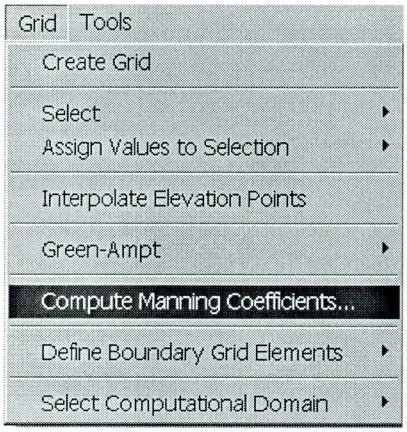
Now click on the *Soil Table* tab and click on the *Import Soil Table*. Select the appropriate soil table file. The data is displayed in the following dialog box:



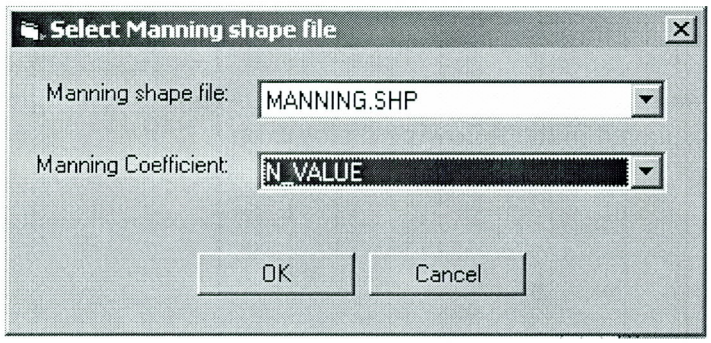
Id	Description	XSAT	Rock outcrop	Eff. Imperv. area
6451	"Antho sandy loams"	0.41	0	100
6452	"Antho gravelly sandy loams"	0.41	0	100
6453	"Antho-Carrizo-Maripo complex"	0.58	0	100
6454	"Antho-Carrizo-Maripo complex, low precipitation"	0.58	0	100
6455	"Anthony sandy loam"	0.43	0	100
6456	"Anthony-Arizo complex"	0.62	0	100
6457	"Anthony-Arizo complex, low precipitation"	0.62	0	100
6458	"Arizo cobbly sandy loam"	0.96	0	100
6459	"Beeline-Cipriano complex, 3 to 45 percent slopes"	0.27	0	100
6531	"Agualt and Ripley soils"	0	0	100
6532	"Agualt and Ripley soils, saline-sodic"	0	0	100
6533	"Ajo-Gunsight-Pompeii complex, 3 to 25 percent slopes"	0.66	0	100
6535	"Carrizo-Dateland complex, 0 to 3 percent slopes"	0.82	0	100
6536	"Carrizo-Momoli complex, 0 to 3 percent slopes"	1.2	0	100
6537	"Cherioni very cobbly fine sandy loam, 3 to 10 percent slopes"	0.4	0	100

The final step is to click on the Compute Green-Ampt button to create the INFIL.DAT file.

Compute Manning Coefficients... Command (Grid Menu)



This command will compute the Manning's n-values based on the information in a roughness shape file. With a project open, first import a Manning's n-value shape file using the *Import Shape File* command on the File menu. When you click on the *Compute Manning Coefficients* command, the following dialog box appears:



Select the Manning shape file name (MANNING.SHP in this case) and the Manning coefficient field (N_VALUE in this case) and click'OK' to calculate Manning coefficients for each grid element. Please note that this data will be written to the FPLAIN.DAT only when you apply the *Create FPLAIN.DAT and CADPTS.DAT Files* command on the File menu.

Define Boundary Grid Elements Command (Grid Menu)

Use this command or the Define Boundary Grid Elements toolbar icon to mark the grid elements that will constitute the boundary of computational domain. These boundary grid elements are not used in a FLO-2D model flood simulation and are not included in either the FPLAIN.DAT or CADPTS.DAT files.

To use the *Define Boundary Grid Elements* command or icon:

1. Select the *Define Boundary Grid Elements* command (Grid menu) or the toolbar icon and click on the left mouse button. The mouse pointer changes to a cross.

2. Select the boundary grid elements. The grid element will change its color to red.

3. Repeat step 2 as many times as needed to select all boundary grid elements.

To undo this operation simply click again on a previously marked grid element. To mark a group of grid elements press the **Shift** key and the left mouse button and then drag the mouse pointer over the desired grid elements. To deselect any group of grid elements, press the **Control** key and with the left mouse button depressed, drag the mouse pointer over the grid elements you want to unmark.

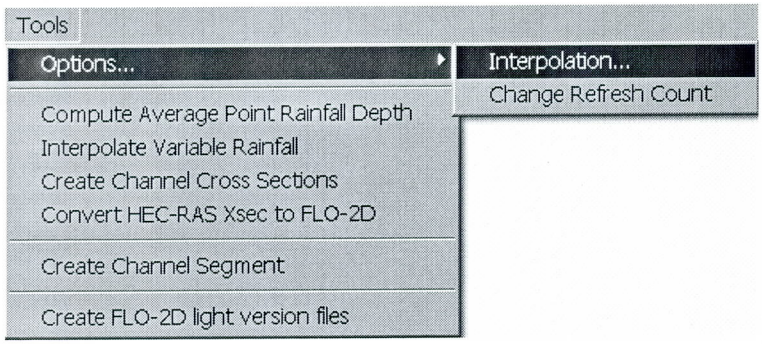
Select Computational Domain Command (Grid Menu)

Use this command to select the computational domain. Once you have finished outlining the flow domain with a closed boundary, you must identify computational domain (potential flow surface).

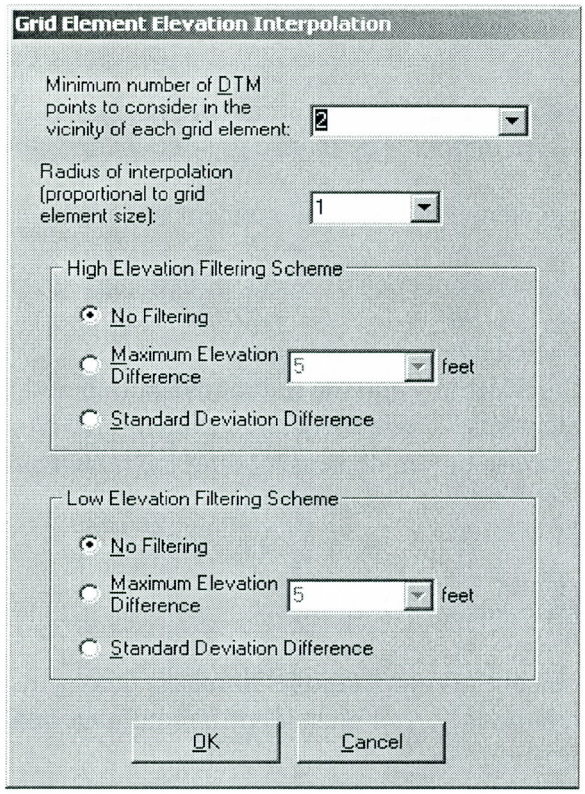
1. Choose the *Select Computational Domain* command. The cursor changes to a cross.

2. Click on any grid element inside of the closed area of the computational domain. If the computational domain area is closed, the system responds with an acceptance message. If the boundary elements do not completely enclose an area, you will be informed that the area is still open. Additional boundary grid elements must then be marked to completely close it.

Options.../Interpolation... Command (Tools Menu)



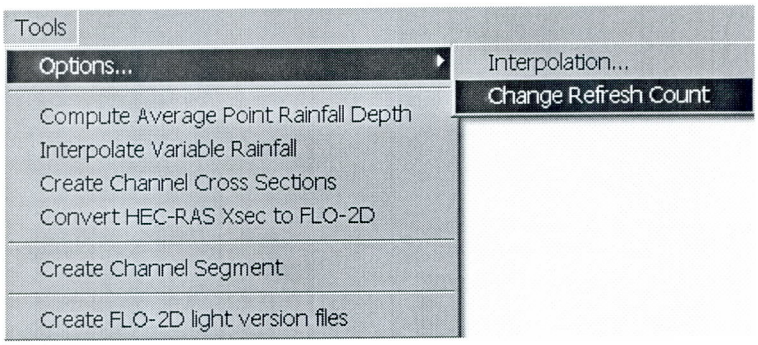
This command defines various options that control the interpolation process. When you select this command, a dialog box appears for the selection of several interpolation parameters:



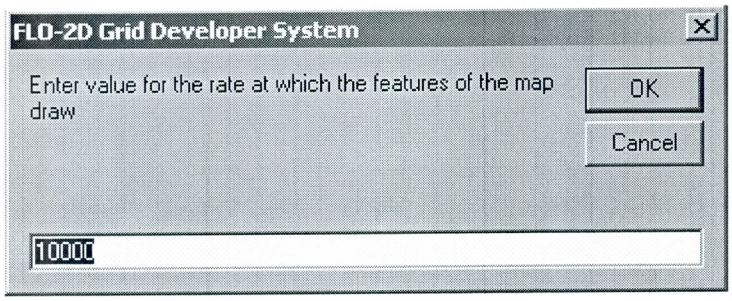
Dialog Box Options:

Option	Description
Minimum number of DTM points to consider in the vicinity of each grid element	To calculate an interpolated elevation from random DTM points that will represent each grid element, the algorithm will use this minimum number of DTM points (or more) closest to the grid element center.
Radius of interpolation (proportional to grid element size)	This radius defines a circle around each grid element center or node based on a multiple of the grid elements size. Only the DTM points inside this radius are considered in the interpolation process. If an insufficient number of elevation points are enclosed, the circle is expanded until at least the minimum number of points is found.
No filtering (High or Low Elevations)	No filtering is performed on the elevation points.
Maximum elevation difference (For either High or Low Elevations)	For this option, the algorithm calculates a mean elevation using all the DTM and any assigned elevation points within the interpolation radius. Then all those points that exceed the selected maximum elevation difference either higher or lower than the mean are discarded and the mean elevation is recomputed and assigned to the grid element.
Standard deviation difference (For Either High or Low Elevations)	The interpolation algorithm calculates the standard deviation of DTM point elevations and then neglects all those points whose elevations are either higher or lower than the standard deviation to determine the mean elevation to represent the grid element.

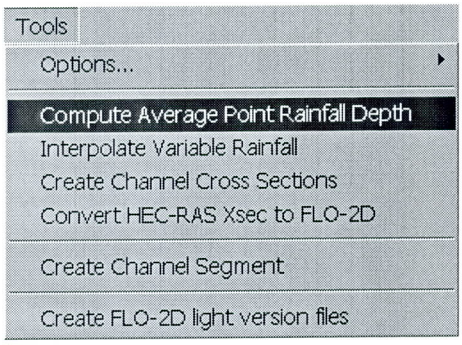
Options.../Change Refresh Count Command (Tools Menu)



Use this command to speed up the redraw or refresh rate for large projects. The refresh count rate specifies how often the screen is updated during the mapping drawing. If there are 1000 vertices to draw and the refresh count is 500, then the screen will be refreshed or updated twice. A higher refresh rate results in faster map draw times while a slower rate provides a smoother draw appearance. When you select this command, a dialog box appears prompting you to give a value for the rate at which features of the map or image are redrawn:

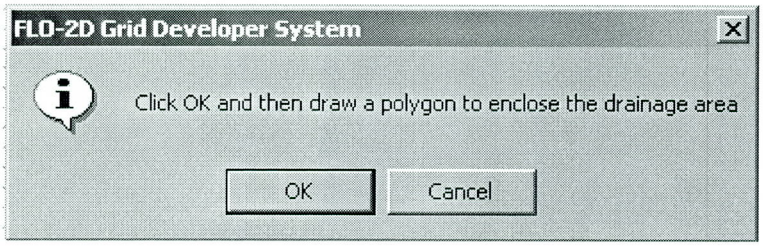


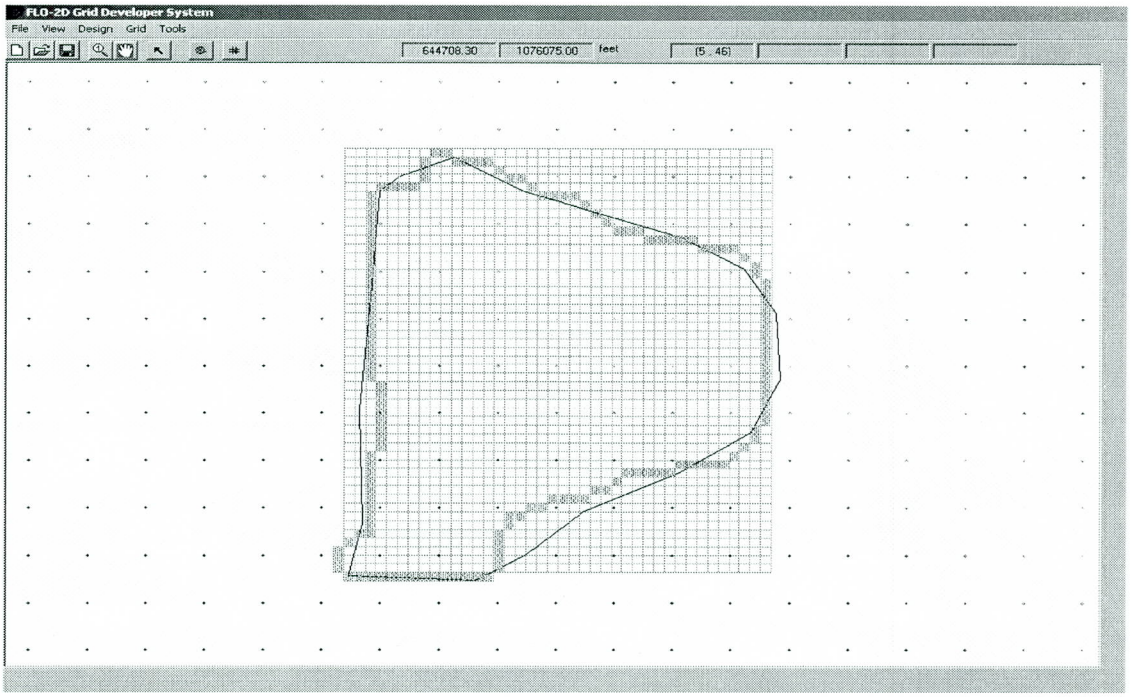
Compute Average Point Rainfall Depth Command (Tools Menu)



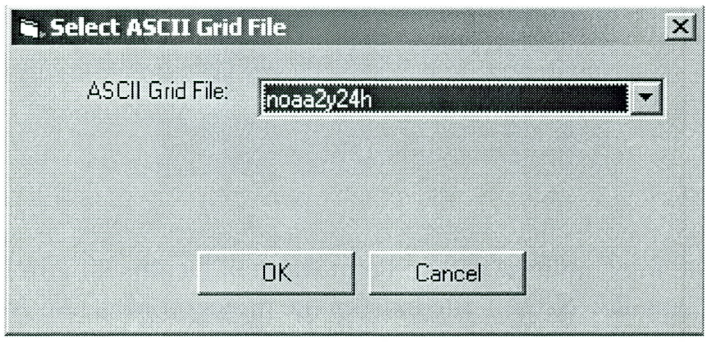
Use this command to compute Average Point Rainfall Depth (RTT in RAIN.DAT file). Note that it is required that a RAIN.DAT file exist in the project folder. An ArcInfo ASCII grid file with rainfall data must be previously imported to the GDS. The format of this file is the

standard ArcInfo ASCII grid with the values corresponding to rainfall depths. The points will be displayed on the screen. Then to compute the average rainfall depth, a polygon will be drawn and only the rainfall points inside the polygon will be used in the calculation.

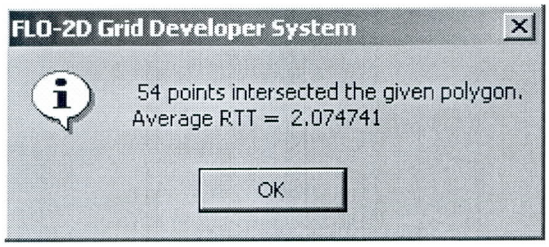




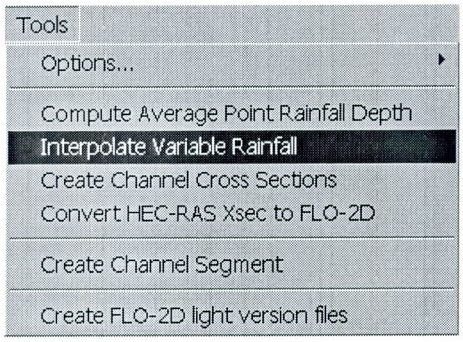
Then choose the appropriate ArcInfo grid file from the file list in the following dialog box:



The following message will identify the number of grid points used to compute the average rainfall depth RTT value. The RAIN.DAT file RTT value will be updated.



Interpolate Variable Rainfall Depth Command (Tools Menu)

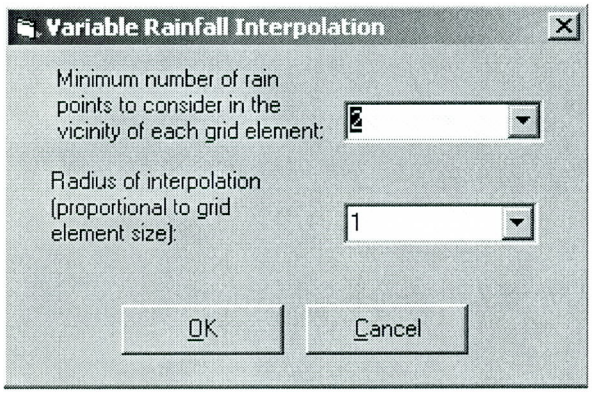


This command will interpolate temporal rainfall for each FLO-2D grid element, based on spatially varied rainfall data in an ArcInfo ASCII grid file format. A file must be prepared that contains the storm beginning and ending dates and times, time interval of the rainfall data files (in minutes), number of reporting storm time intervals (one rainfall data file per interval) and the file name and location. Each subsequent rainfall data file will list the rainfall depth at a rain gage grid location (rainfall grid not the FLO-2D grid system) at that specific time. For example:

6/12/2002 13:00 6/12/2002 14:00 60 2
C:\Models\FLO2D_2002\Maricopa2002\Raindata2\rain13h.dat
C:\Models\FLO2D_2002\Maricopa2002\Raindata2\rain14h.dat

In this case, the beginning date of the storm is 6/12/2002 and the storm beginning time is 13:00 hours. The storm ending date and time is 6/12/2002 and 14:00 hours. The time interval between rainfall data files is 60 minutes and there are two rain grid files in the list. The rainfall file format is the standard ArcInfo ASCII grid with the values corresponding to rainfall depths. Each file corresponds to a rainfall depth at a specific time for rainfall grid system. The rainfall grid file (e.g. NEXRAD data) is prepared outside the FLO-2D software system on the basis of the rainfall gage distribution. This rainfall distribution over the rainfall grid has to be transformed

into spatially varied rainfall assigned to grid elements in the FLO-2D grid system. This is accomplished using the variable rainfall interpolation algorithm in the GDS program. Once the ArcInfo ASCII file is read, the following dialog box permits adjustment of rainfall grid interpolation parameters.



Option	Description
Minimum number of rain grid points to consider in the vicinity of each grid element.	To calculate the interpolated rainfall depth for each FLO-2D grid element, the algorithm uses at least this minimum number of points closest to the grid element.
Radius of interpolation (proportional to grid element size).	This radius defines a circle around each grid element based on a multiple of the grid elements size. Only rainfall grid points inside this radius are considered in the interpolation process. If the number of rainfall grid points is insufficient, then the circle is automatically enlarged until at least the minimum number of points is found.

After the interpolation is complete, choose a RAINCELL.DAT file location to save results. The RAINCELL.DAT file is then used by the FLO-2D model to simulate the rainstorm. The IREALRAIN variable switch in the FLO-2D RAIN.DAT file has to be set to 1 (on) to simulate a real-time storm using the RAINCELL.DAT file.

The format of the RAINCELL.DAT is as follows:

Line 1. Beginning date and time, ending date and time, time interval (in minutes) and number of time intervals.

Line 2. Grid element number and the total rainfall depth at that time. This line is repeated for each FLO-2D grid element.

Successive rainfall data for the entire grid system follows for the number of time-interval groups. For the example above, there would be two sets of rainfall data for the entire grid system.

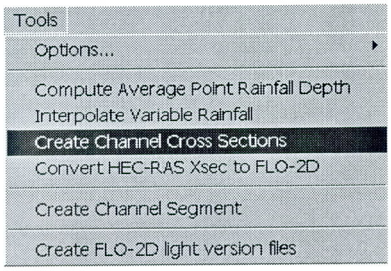
6/12/2002 1:00:00 PM 6/12/2002 2:00:00 PM 60 2

```
1       1.8538
2       1.8537
.               .
.               .
.               .
1234    2.272
1235    2.2709

1       1.4794
2       1.4791
.               .
.               .
.               .
1234    1.6294
1235    1.6286
```

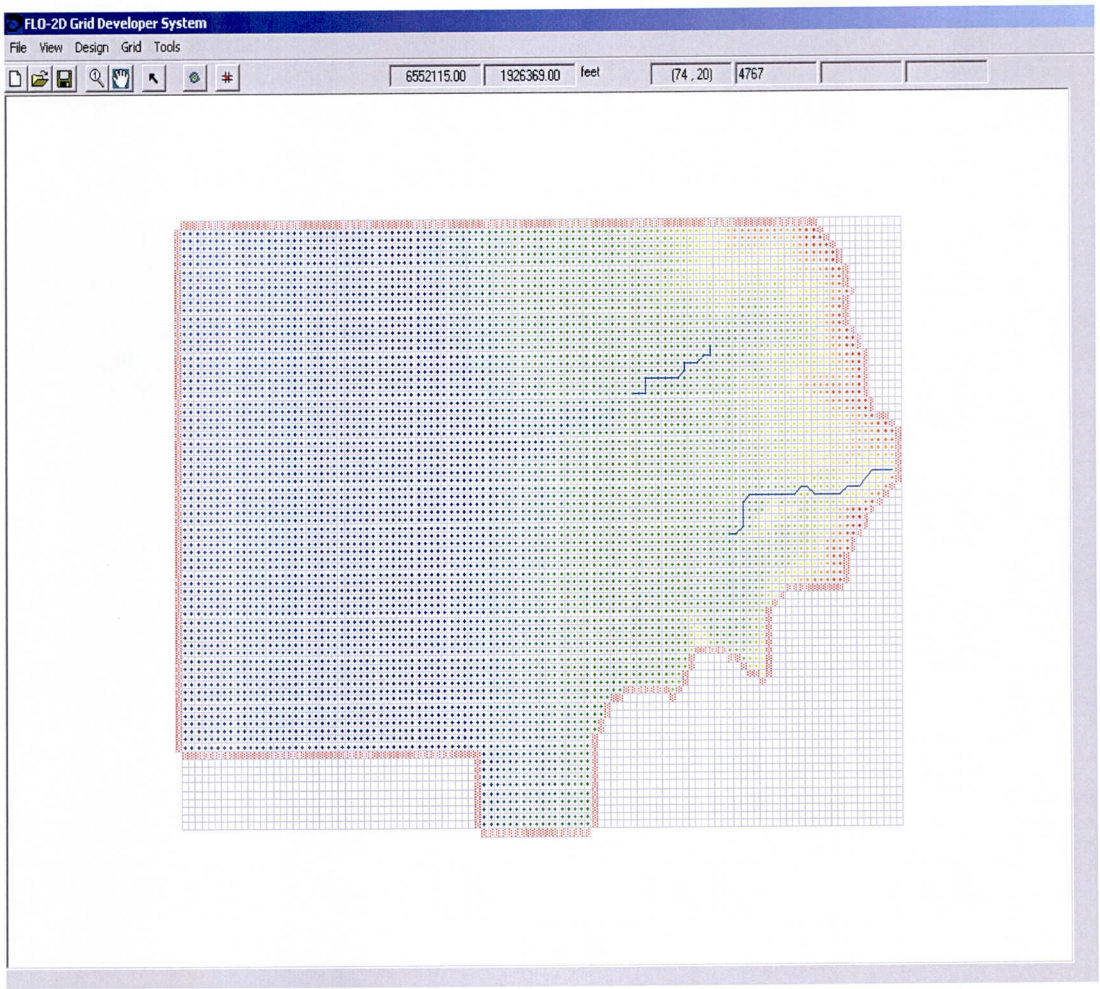
In this above example the beginning date is 6/12/2002 the beginning time 1:00 PM, the ending date 6/12/2002 and the ending time 2:00 PM. The time interval between files is 60 minutes, there are two time intervals and 1235 grid elements. The rainfall depth for grid element number 1 at time 1:00 PM is 1.8538 in. Successive grid element rainfall depths follow. For the second time interval, the rainfall depth for the first grid element is 1.4794 in and the rest of the grid elements follow. If the grid system has been prepared in International Units (meters), then the rainfall data file should be prepared in mm.

Create Channel Cross Sections Command (Tools Menu)

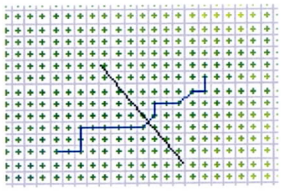


With this GDS component, you can extract channel cross sections from existing DTM data and assign the cross sections to the FLO-2D XSEC.DAT data file. The CHAN.DAT files and XSEC.DAT files have to be prepared in advance. The FPLAIN.DAT file associated to CHAN.DAT file must also be prepared. In the current GDS version, you **cannot** add new channel elements to an existing FLO-2D model with this GDS cross section feature. You can only replace existing cross sections. A simple approach is to create the CHAN.DAT and XSEC.DAT files using any existing cross section survey data. If no cross section data exists, then use one generic cross section for all the channel elements. Then new cross sections can be cut from the digital terrain model to replace the existing cross sections in the XSEC.DAT file. The procedure to accomplish this is as follows:

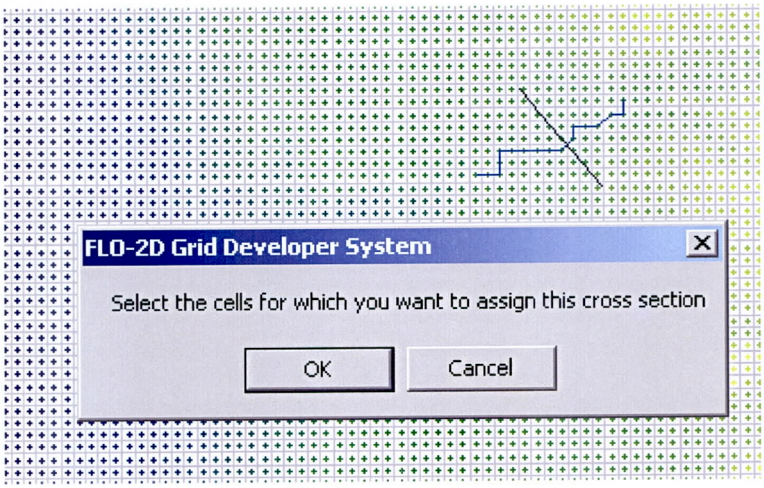
1. First the channel segments must be retrieved as explained in the *Import Channel Segments* section. The channel segments will be displayed on the screen. The colored points indicate the DTM point elevations. For example:



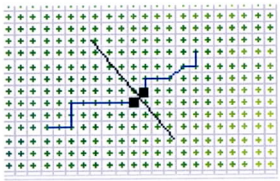
2. Next draw the channel cross section on the digital terrain map. First, zoom-in on a channel reach, then click with the mouse for the first point and double-click for the second point. The cross section line will be plotted.



3. Now, select the channel grid elements for which you will assign this cross section data.



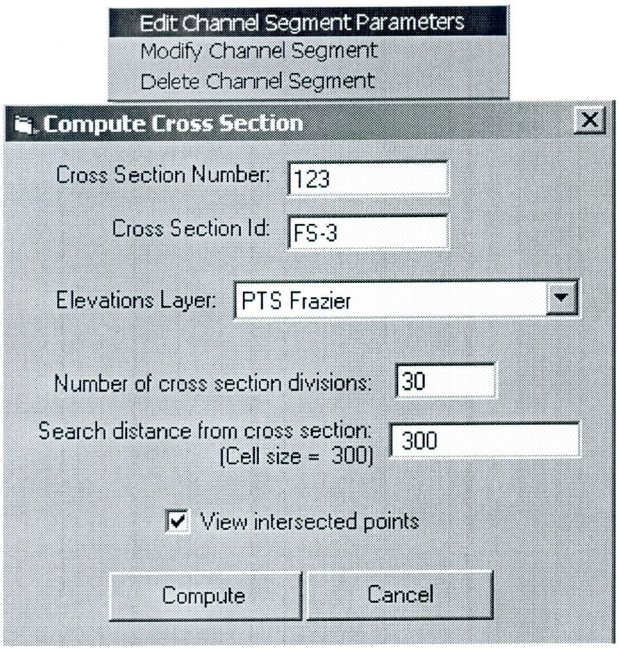
4. Double click on the last grid element to end the selection process. The selected grid elements will be marked.



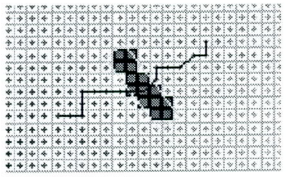
5. Set the desired cross section number and ID, the number of cross section divisions and the search distance for DTM points (perpendicular) from the selected cross section. The GDS will search all DTM points contained in this area and use them to compute the cross section points.

Edit Channel Geometry and Segment Parameters.

To edit channel geometry parameters, click on an a channel segment. A submenu appears with several options:

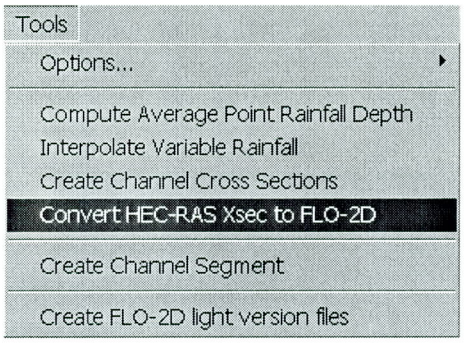


6. Click on *Compute* and GDS will show the computing area, calculate the cross section points, add the cross section data to the XSEC.DAT data file and update the previously selected channel grid elements with the new section in the CHAN.DAT data file.



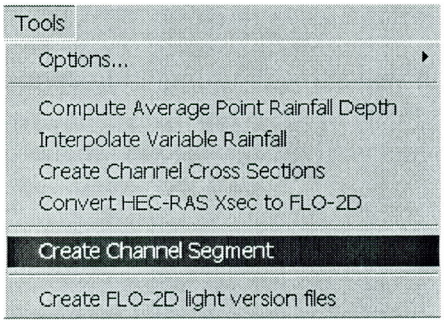
After all the cross sections have been extracted from the digital terrain model, you should review the channel profile in the PROFILES program. In PROFILES you can adjust the channel bed profile, edit the cross sections and interpolate between surveyed cross sections.

Convert HEC-RAS Xsec to FLO-2D Command (Tools Menu)



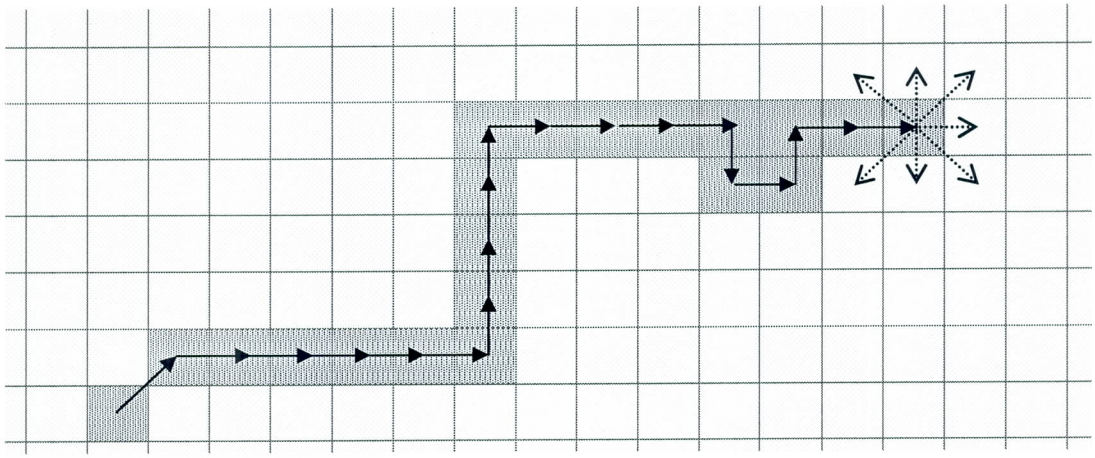
This feature will read cross section data available in HEC-RAS format, convert it to FLO-2D format and save it to XSEC.DAT files. The program reads the file, locates the cross section key words, extracts the x, y, z data from each cross section and calculates the new cross section in the required FLO-2D as specified in for the XSEC.DAT file. When this process is completed, another file dialog box appears to let the user chose the location of the output XSEC.DAT file.

Create Channel Segment Command (Tools Menu)



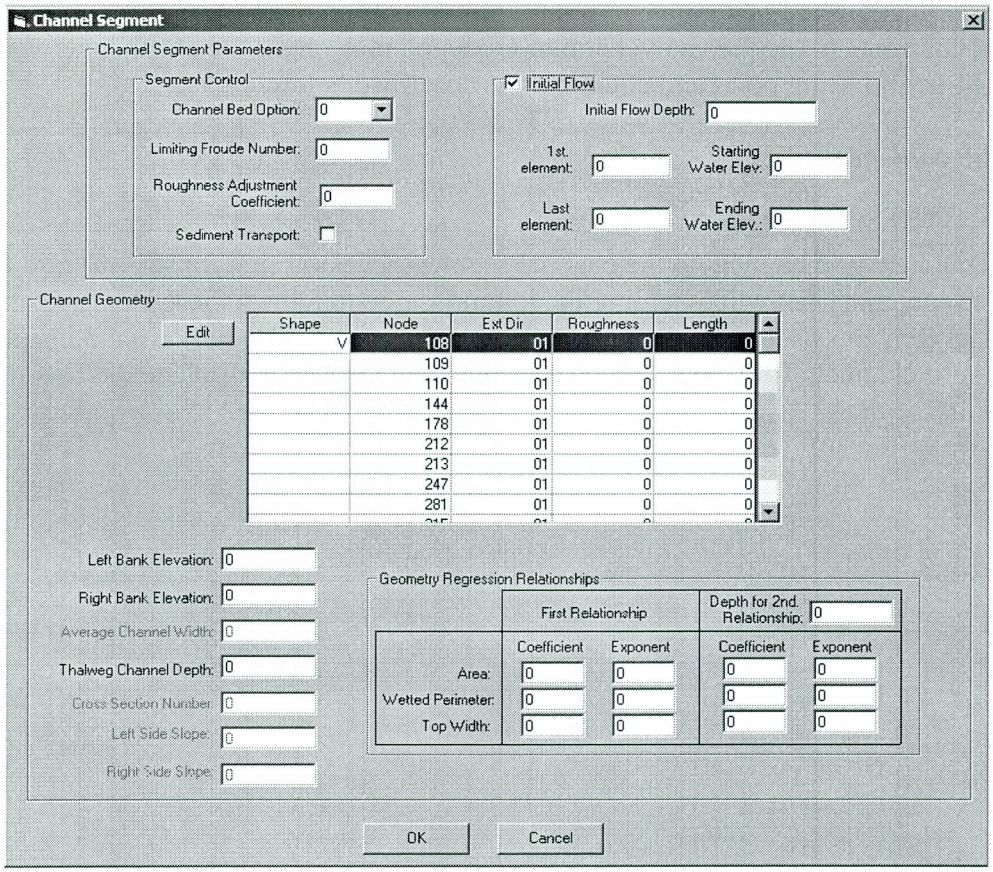
The FLOENVIR processor program was designed to graphically edit the spatial variability of the floodplain and channel attributes including floodplain elevation and n-values, channel geometry, levees, streets, inflow and outflow nodes, and area and width reduction factors. These functions are being transferred to the GDS program because of improved graphics and windows controls. In the present version of GDS you may create the channel network in an existing FLO-2D grid using point and click tools as described below:

1. On the *Tool menu* click *Create Channel Segment*
2. The mouse pointer changes to a cross.
3. To add a new grid element to the channel segment, click on first (upstream) grid element of the segment. Then while holding the left mouse button down the system selects channel elements as the user moves over the grid system. Use background images to locate the channel. Channel elements are joined by arrows to visualize the channel location:



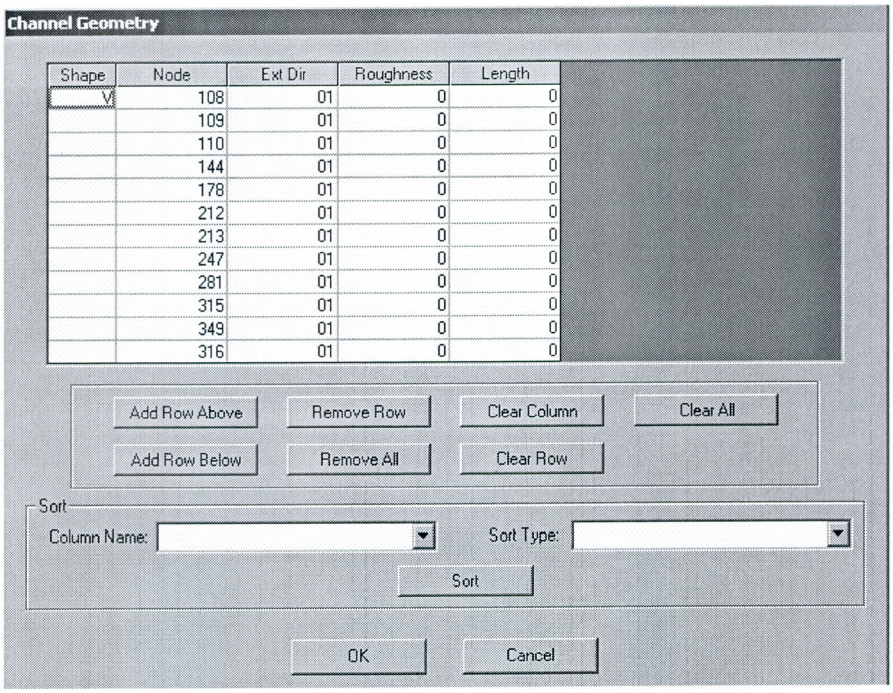
As the channel development progresses from an element to the next, only the 8 adjacent elements can be accessed (see above figure). When the mouse pointer is near the center of the next element to be selected, a solid arrow appears to indicate that that element is the next one in the segment. The user can backup the pointer over the segment to unselect channel grid elements. No clicks are necessary, the users only needs to sweep the pointer over the desired elements. The operation finishes when the user double-clicks with the left mouse button.

Select *Edit Channel Segment Parameters* and the Segment Parameters dialog box is displayed:



Notes:

- Each row in the "Channel Geometry" table corresponds to a channel grid element.
- The "Geometry Regression Relationships" data table is enabled only when a grid element of type "V" is selected in the "Channel Geometry" table.
- Depending on the channel element data base some columns may be disabled.
- The editing of the segment channel elements is achieved by pressing the "Edit" button. The following dialog window appears:

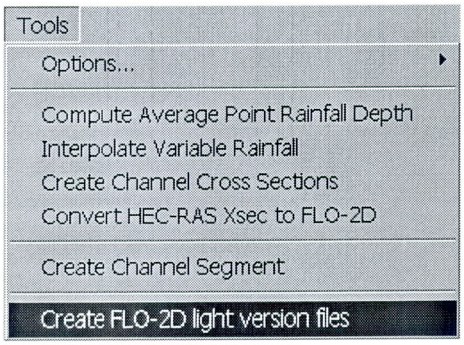


- Interact with the table to edit the channel attributes.

Delete Channel Segment

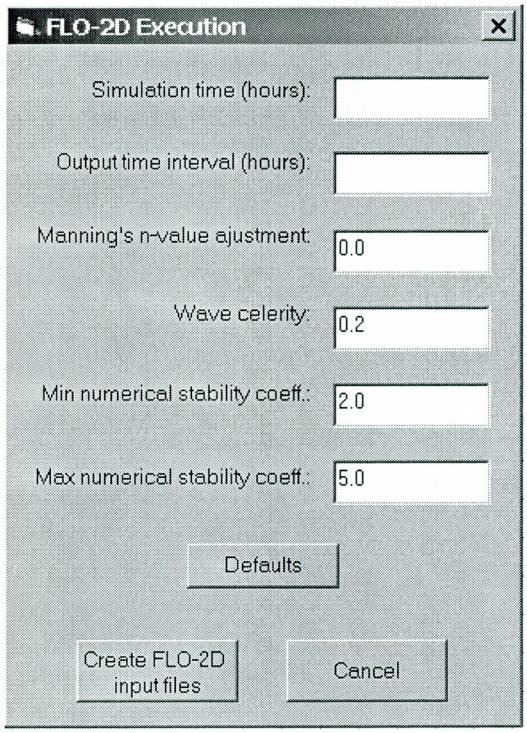
This command will delete the selected channel segment.

Create FLO-2D Input Light Version Files Command (Tools Menu)



Using this command the system will create the necessary FLO-2D files to start a simulation with only the minimum essential data. The *FLO-2D Light* option includes functions to automate the creation of the essential files to perform an overland flood simulation without any components. These files will be quickly generated, the FLO-2D simulation initiated and the results presented using the MAPPER post-processor program. To use the *FLO-2D Light* option, the following steps must be completed:

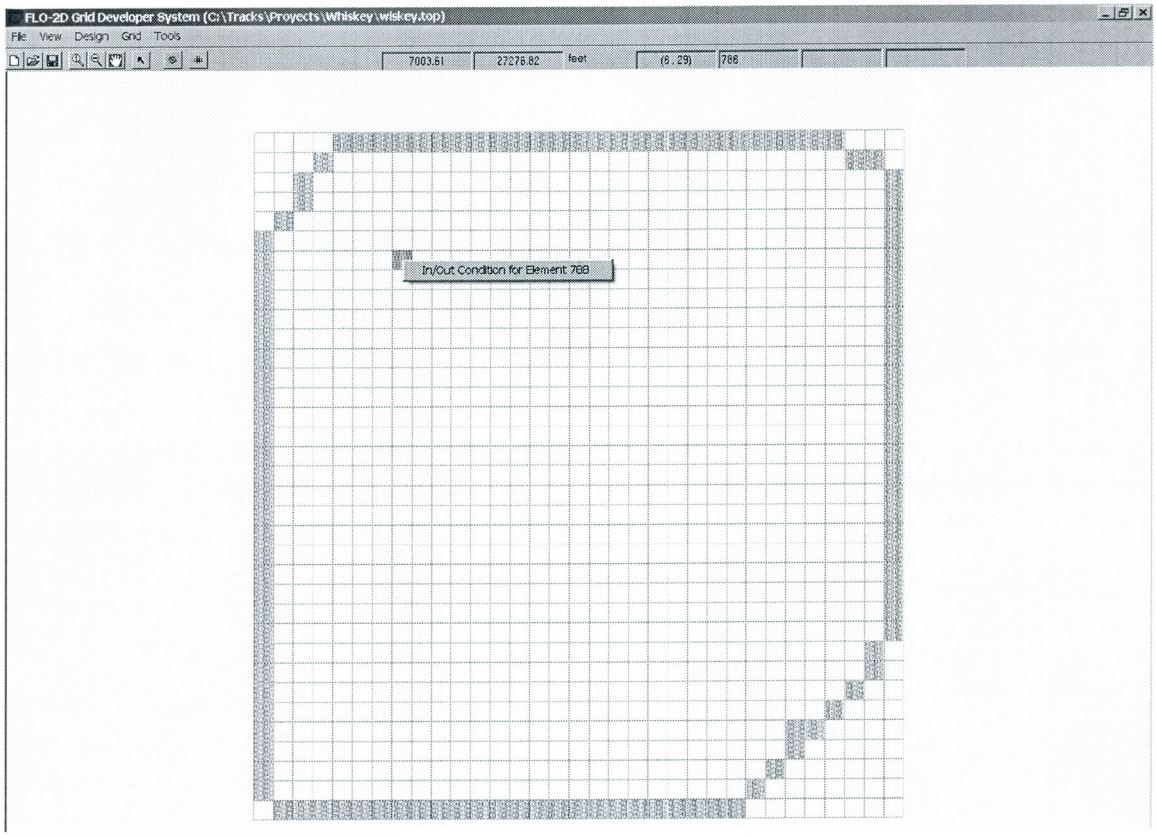
- Create a working area with the particular region limits (this may include importing the DTM points). Background images or aerial photographs can be imported to delineate region boundaries;
- Designate the boundaries of the project site or flow domain. The GDS will automatically generate this boundary;
- Input the terrain data or interactively generate topographic contours (you may input the DTM data at the outset);
- Interpolate the digital terrain elevation data and assign FLO-2D grid element elevations;
- Define inflow nodes and assign hydrographs to these grid elements;
- Define outflow nodes;
- Click on the *Select Create FLO-2D light version files* on the *Tools* menu (see above dialog box) to input the following parameters:



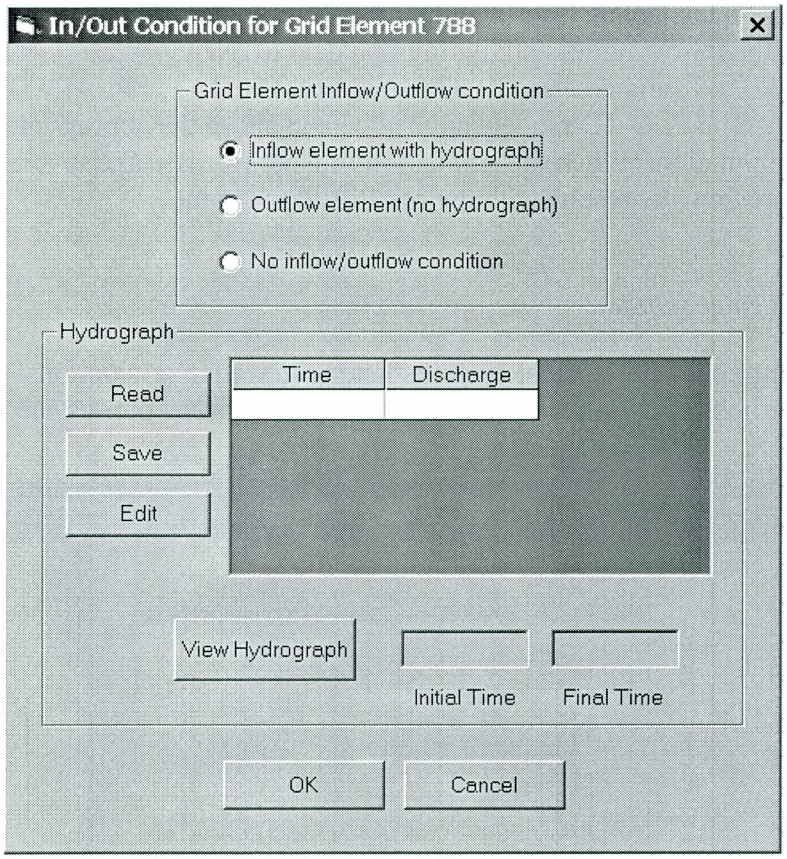
When the user clicks on the Create FLO-2D input files, the system checks that all conditions are correctly set, writes the inflow nodes and hydrograph data to INFLOW.DAT, writes the outflow nodes to OUTFLOW.DAT, and creates CONT.DAT, TOLER.DAT, FPLAIN.DAT and CADPTS.DAT files. The FLO-2D model can then be and chose the appropriate CONT.DAT file to perform the simulation. When the simulation finishes (it may take several minutes or hours), the user can review the results with the MAPPER program.

Hydrographs Management Option

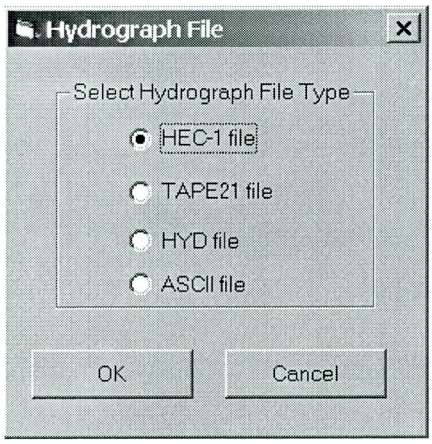
Inflow hydrographs will be created, edited and reviewed in this option. To set an inflow node, the user right-clicks on a grid element and the system shows a pop-up menu:



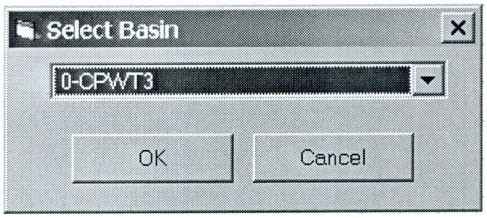
User selects "*In/Out Condition for Element nnn*". The system displays the following dialog box (in this case for grid element 788):



When the user selects the radio button *'Inflow element with hydrograph'*, the 'Hydrograph' input frame is activated (enabled). The "Read" button displays a dialog box to chose from HEC-1, Tape21, HYD or ASCII files:



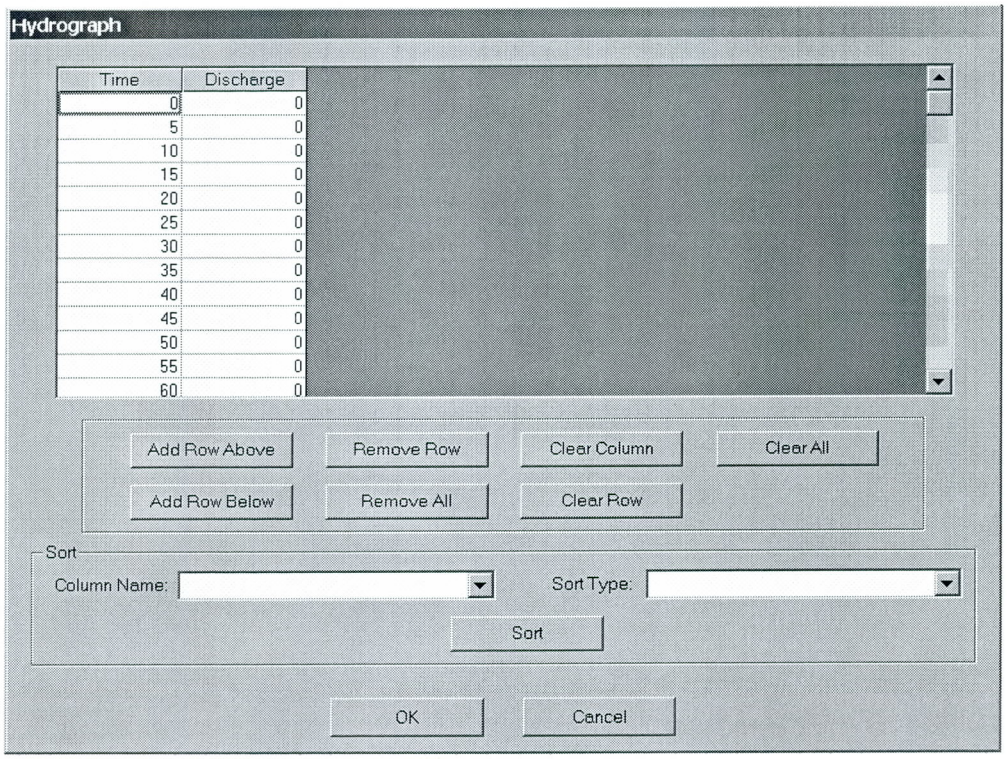
- The HEC-1 file option will present a series of hydrograph basin names from a HEC-1 model output file from which the user should select an inflow hydrograph:



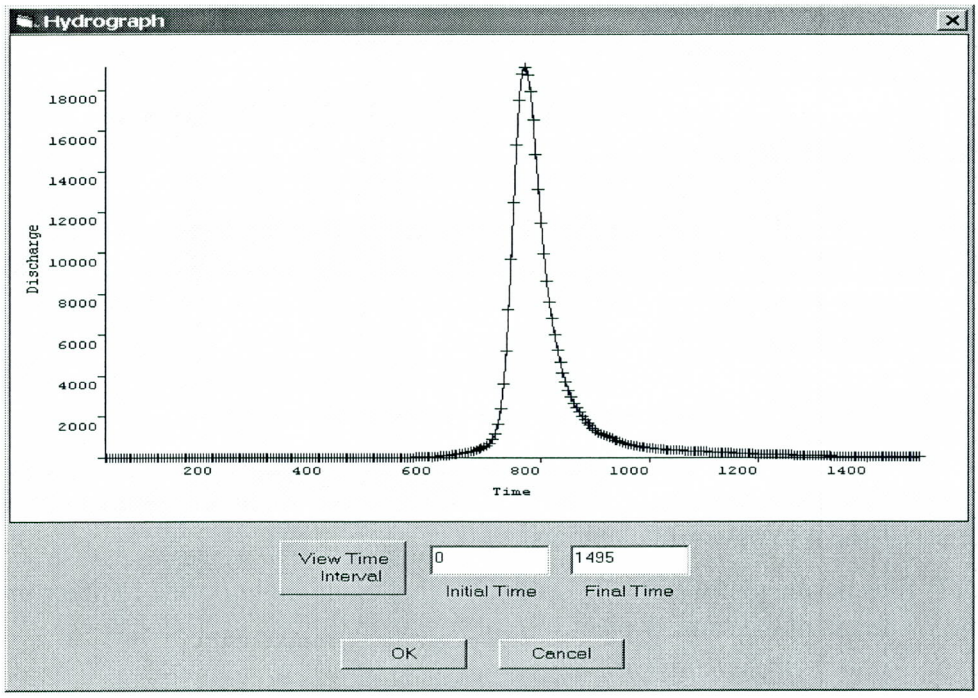
After selecting hydrograph (a table) and clicking 'OK' the hydrograph data are loaded into the above dialog box.

- HYD files are ASCII files used to store hydrographs. They are created with the "*Save Table*" button. The first line contains the initial and final time of the hydrograph selected with the "Select Time Interval" button. Two columns of data follow; time and discharge. These files will be valuable to:
  - Recover the hydrographs when a project is read from a TOP file;
  - To redefine the project area when the INFLOW.DAT becomes obsolete because the grid element numbers change. The HYD files enable the hydrographs to be recovered for assignment to new grid elements.
  - HYD files permanently stores hydrographs that could be imported to other projects or grid elements in the same project.

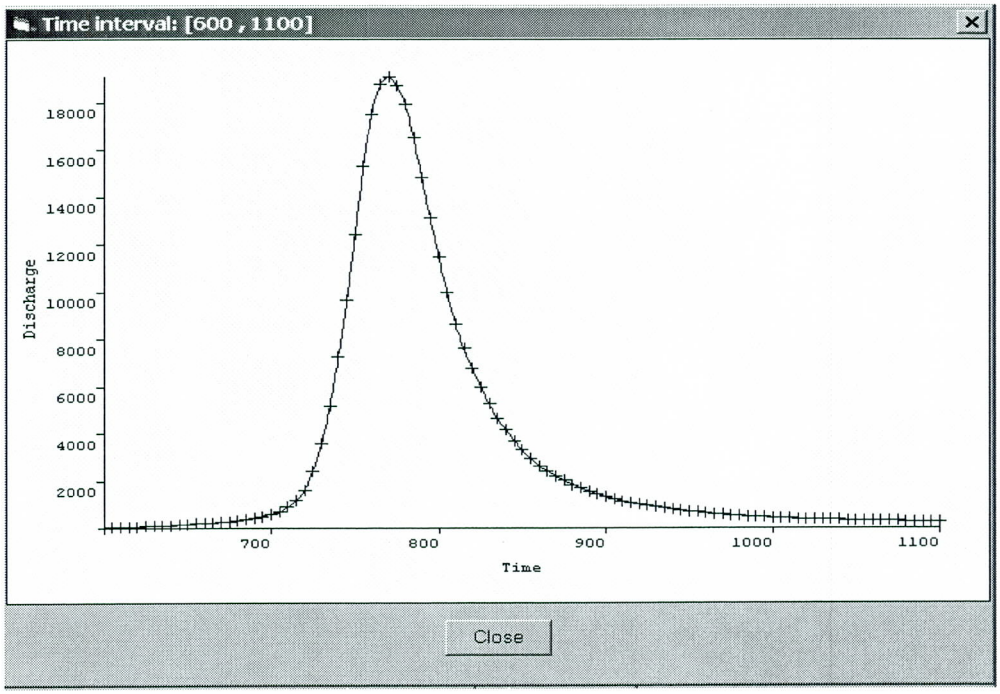
The "*Edit Table*" button can be used to edit hydrograph values, insert rows, delete rows, or sort rows in ascending order (time column). This button a displays an editor dialog box:



The "*View Hydrograph*" button of the above In/Out Condition dialog box plots the hydrograph in the following window:



The user may select the Initial and Final time for the inflow hydrograph. For example, selecting Initial time = 600 and Final time = 1100 will redefine the hydograph limits.



Inflow grid elements with assigned hydrographs are displayed in GDS with a distinct color to for identification. When the user selects "*No Inflow/Outflow condition*" and clicks "OK" the cell recovers its original color to indicate the absence of a hydrograph. Inflow nodes and the linked hydrographs will be written to the INFLOW.DAT file. Outflow nodes are assigned though the option "*Outflow element (no hydrograph)*". The list of outflow nodes will be written to the OUTFLOW.DAT file.

Initial Assignment of Hydrographs

The initial assignment of hydrographs to inflow nodes has three options:

1. ***Project is created from FLO-2D project*** (GDS command "*File. New Project. from FLO-2D Project*"). In this case INFLOW.DAT is used by the system to assign hydrographs to pre-selected grid elements.

2. ***Project is read from TOP*** (GDS command "*File. Open Project*"). The new TOP will include references to HYD files that are related to grid elements defined by the user.

3. Other forms of creating a project require starting the project from scratch. The user will be required to create the grid system and then later right-click on the selected inflow nodes to assign hydrographs as described above.

3.4 PROFILES

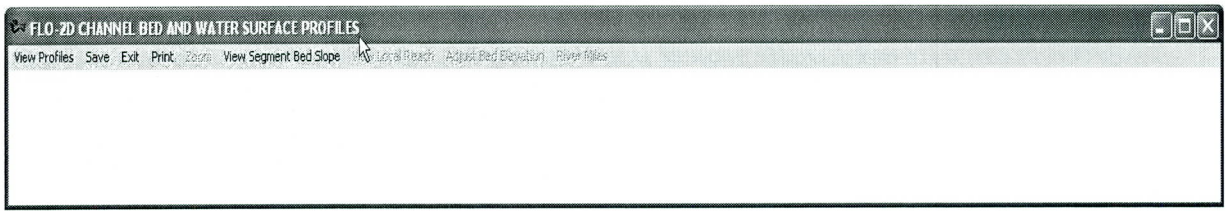
The PROFILES processor program displays the channel slope and permits interactive adjustment of the channel bed elevation or channel depth. It will also display the channel cross section geometry and interpolate the slope and cross section geometry between survey cross sections. PROFILES can also be used to view output water surface profiles (see the Post-Processor Programs Section). To run PROFILES, access the program in the GUI pre-processor pull-down menu or double click on the PROFILES.EXE in the project subdirectory in Explorer. Make sure that PROFILES.EXE is in the project subdirectory.

Before using the PROFILES program, the CONT.DAT, CHAN.DAT and FPLAIN.DAT files have to be created. The XSEC.DAT will also have to be created for the channel cross section data option. The format for these files will be presented later in the manual. The procedure for using the PROFILES program is as follows:

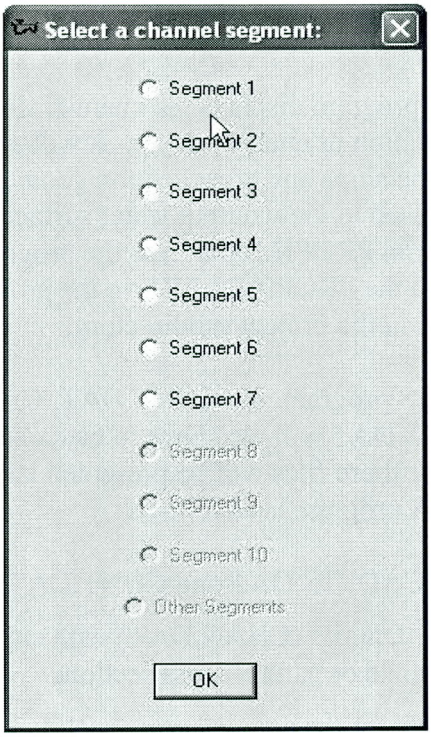
1. Create the FPLAIN.DAT, CADPTS.DAT and CONT.DAT files.
2. Complete the cross section data file (CHAN.DAT) with either rectangular, trapezoidal, channel geometry relationships or natural cross sections.
3. Develop the XSEC.DAT file for the channel cross section option.
4. Run the PROFILES program from the GUI or Explorer.
5. The model bed slope can be compared with surveyed bed elevations by developing the WSURF.DAT file.

6. Upon exiting the PROFILES program, two new files FPLAIN.NEW and CHAN.NEW will be created that contain both the original and adjusted bed elevations and channel depths. You will be prompted to save your data by replacing the original FPLAIN.DAT and CHAN.DAT files with the newly created FPLAIN.NEW and CHAN.NEW files. If you choose not to replace the files, the two files will be available for review.

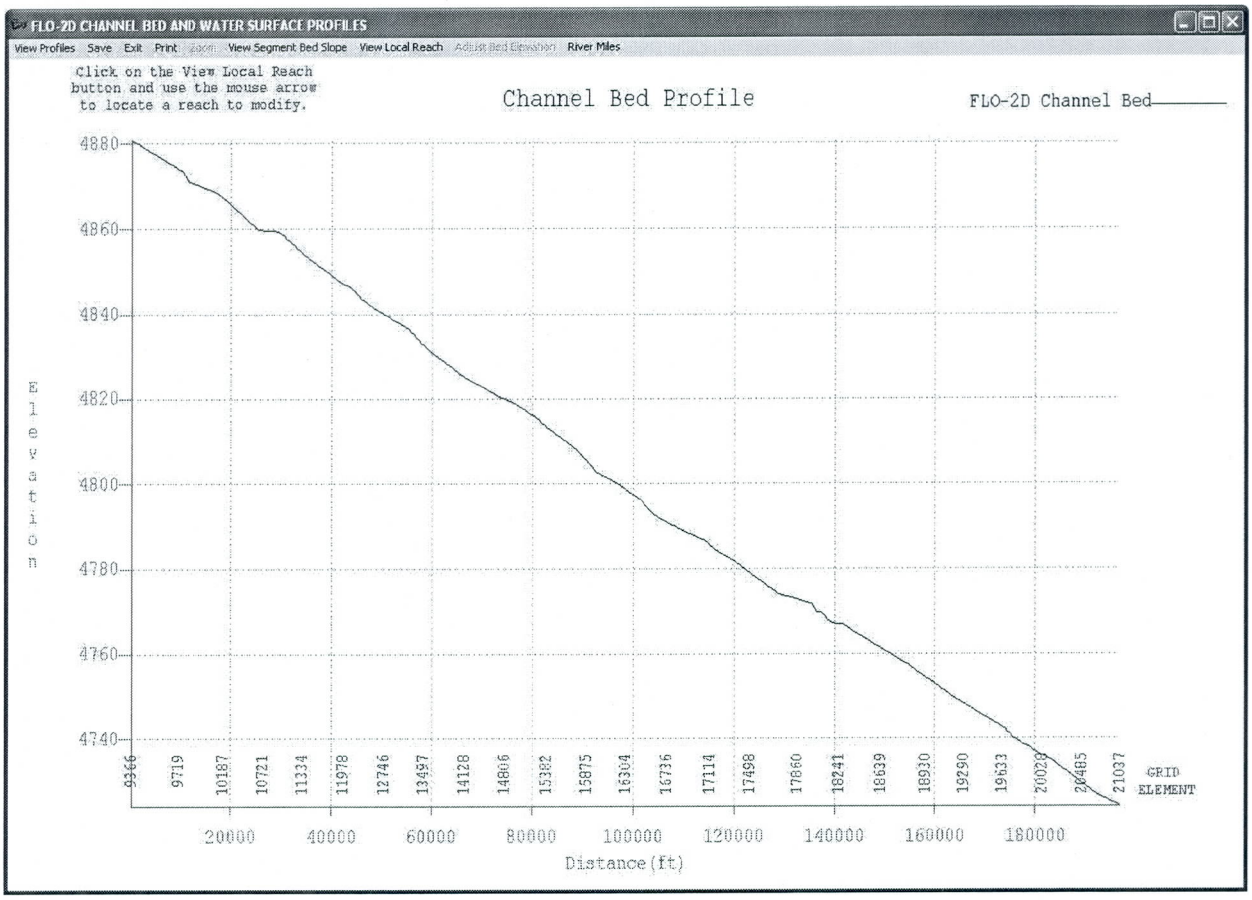
Initially the PROFILES program will display a blank screen with a Main Menu showing an option to 'View Segment Bed Slope'.



Click on 'View Segment Bed Slope' to display a dialog box of the channel segments. If there is only one channel segment in the model, the channel bed slope will be automatically displayed without the dialog box.

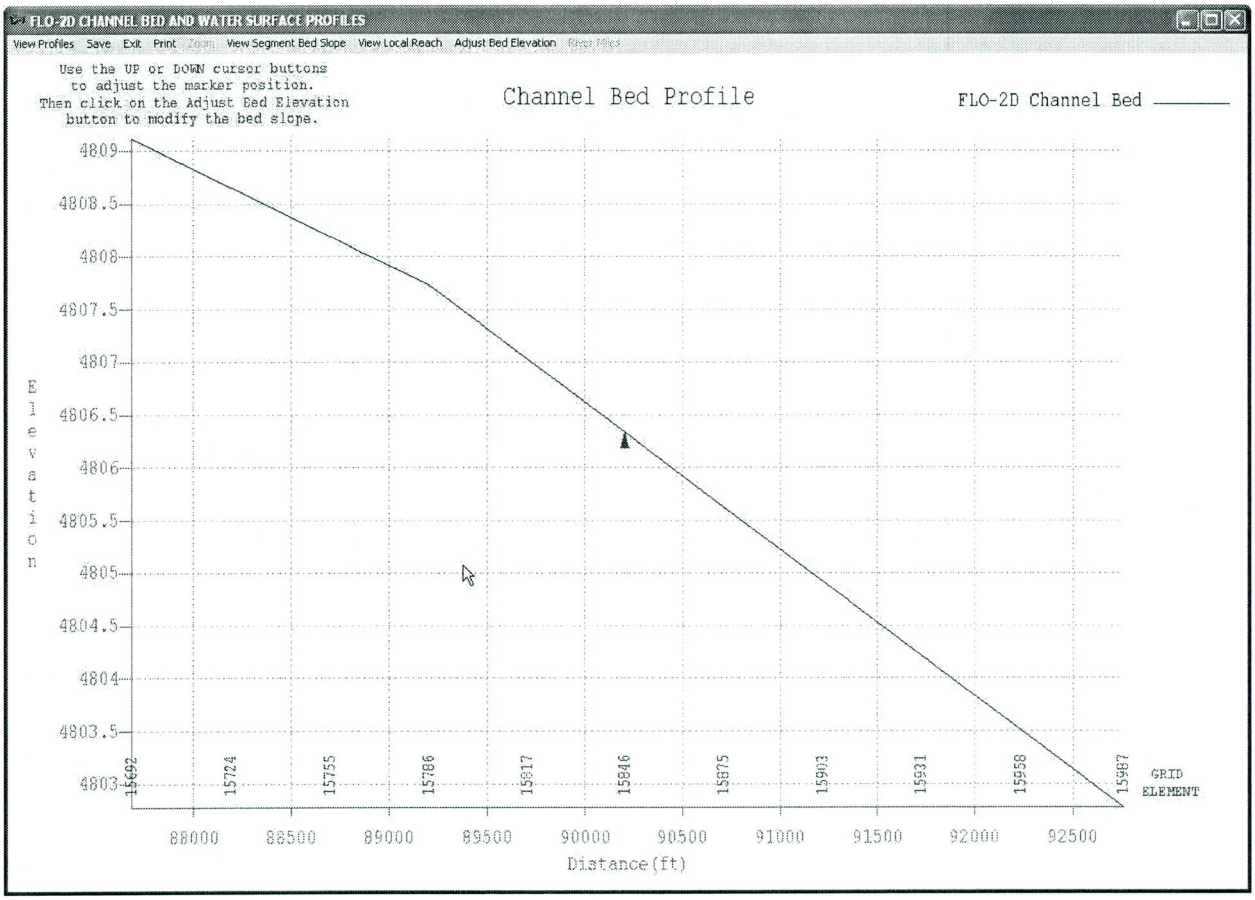


Select a channel segment to display its bed slope and click 'OK'.

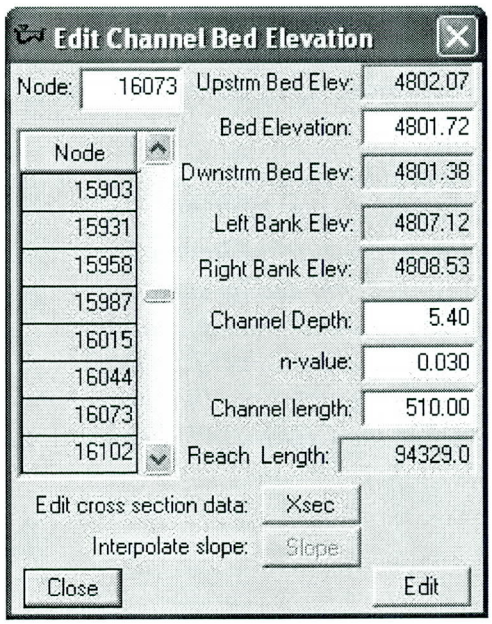


Click on 'View Local Reach' on the Main Menu and point the arrow at a location on the bed profile to display a more detailed profile extending over ten channel elements. The 'Up' and 'Down' cursor keys can be toggled to move up or down the channel.

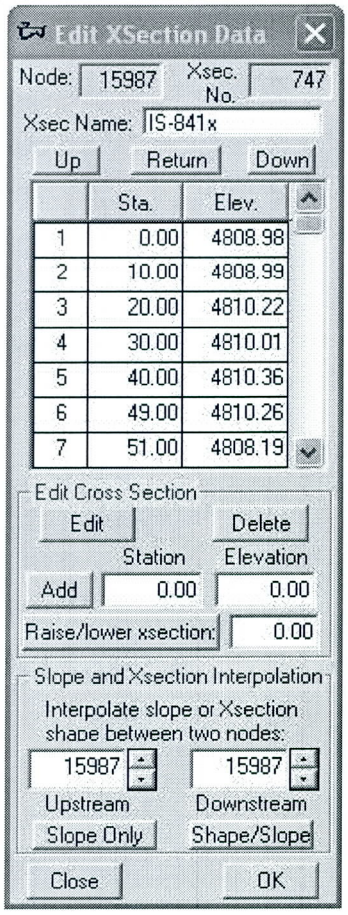
IMPORTANT NOTE: *The channel reference elevation for determining the bed slope is specified by the ICHDEP variable in the CHAN.DAT file. The reference elevation is either the floodplain elevation in the FPLAIN.DAT file or the lowest bank elevation in the CHAN.DAT file. The channel bed elevation (thalweg) is computed by subtracting the channel depth from either the reference floodplain elevation or the lowest bank elevation. A third method of assigning the channel bed elevation is to set floodplain elevation (FPLAIN.DAT file) as the channel bed elevation and recompute the floodplain elevation for that grid element by adding the channel depth to the channel bed elevation. See the CHAN.DAT file description for further discussion on the channel bed elevations.*



The 'Adjust Bed Elevation' mode will display a dialog box in which the bed elevation can be edited. Again, the 'Up' and 'Down' cursor keys can be toggled to move up or down the channel.

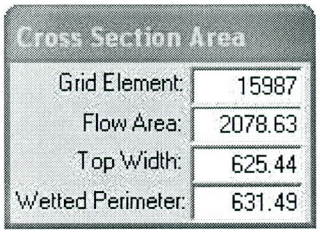


To edit the cross section data, prepare the XSEC.DAT (see the XSEC.DAT file description later in the manual) and click on the 'XSEC' button in the dialog box. The cross section data for each grid element is displayed in a dialog box. The cross section data can be edited using the 'EDIT', 'ADD' or 'DELETE' buttons. Click on the 'UP' and 'DOWN' buttons to move upstream or downstream.



A popup window shows the cross section plot that can be expanded to full view.

A second dialog box displays the computed channel geometry.



Between surveyed cross sections, the slope and the cross section geometry can be interpolated by clicking on the appropriate interpolation buttons. The cross section and slope interpolation are key methods to create average channel conditions in a reach. Usually one cross section will be assigned to several grid elements based on number of surveyed cross sections in a reach. This creates a stair step pattern in the cross section profile. To smooth out the profile, select the grid elements with the surveyed data and use the slope interpolator button 'Slope Only' in the Edit Cross Section dialog box shown above.

The cross section interpolator button 'Shape Slope' in the Edit Cross Section dialog box shown above will interpolate the cross section geometry to those channel elements between the two selected channel elements with surveyed cross sections. This routine provides a method for creating transitional cross sections between wide and narrow channel reaches. The interpolator routine first computes a linear estimate of the channel width based on the channel length between the grid elements. Then each surveyed cross section is projected onto the new width and each station elevation based on interpolated bank reference elevation is averaged. After the cross section interpolation is complete, the channel bed slope is linearly interpolated.

To compare the surveyed bed slope or water surface slope with the model bed slope or the predicted water surface elevation, the WSURF.DAT file must be created. The ASCII file format is:

Line 1. Number of grid elements with a surveyed water surface or bed elevation.
Line 2 to end. Grid element, measured water surface, surveyed bed elevation.

A partial example WSURF.DAT file follows:

```
  100
23759  4651.00   4647.79
23758  4650.10   4646.20
23900  4649.40   4644.30
24004  4648.75   4644.20
24100  4647.80   4643.25
24120  4647.15   4642.60
24130  4646.80   4642.20
24138  4646.45   4641.90
24146  4646.00   4640.90
         ...
```

IV. INPUT FILE DESCRIPTION

4.1 General

The FLO-2D data file formats are described in the following sections. These files are called directly by the model during a flood simulation. For each data file a list of the variables, a partial example data file and an alphabetical description of the variables are presented. Some instructional comments follow to clarify the variable format. Either the GUI or any ASCII text editor can be used to create or edit the data files. The FLOENVIR will also graphically edit some of the data files.

All of the data entries are in free format requiring only a space to separate the variables. The variables are listed in this manual line by line. Each line may contain several variables that are highlighted by bold text and capital letters. Array variables are indexed as shown in the following example from the INFLOW.DAT file, Line 3:

3 **HYDCHAR = 'H', HP(I,J,1), HP(I,J,2), HP(I,J,3); I = 1, NOHYDRO; J = 1, Number of inflow hydrograph pairs**

where:

I and J are array indices;
H(I,J,1) is the time (hrs) at the start of each discretized interval in the channel inflow hydrograph;
H(i,J,2) is the discharge (cfs);
H(I,J,3) is the sediment concentration by volume for a mudflow inflow hydrograph.
HDYCHAR is a line identifier character 'H'
NOHYDRO is the number of channel inflow boundary grid elements;

Line 3 represents a discretized inflow hydrograph that is repeated for each of the hydrograph pairs for each inflow grid element. The data that is represented by Line 3 for the first four time steps is as follows:

Time (hrs)	Discharge (cfs)	Sediment Conc. 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 in the CONT.DAT file. The backup files are useful for the purpose of locating data errors.

4.2 List of Program Files Units

The following list of files indicates the file's unit numbers assigned by the FLO-2D file. These unit numbers may be useful when interpreting data file read errors during runtime.

File No.	File Name	File No.	File Name	File No.	File Name
2	BASE.OUT	40	CONT.BAC	74	CHNBEDEL.OUT
7	ERROR.CHK	41	FPLAIN.BAC	75	STRELEV.OUT
8	Used	42	RAIN.BAC	76	used
9	TOLER.DAT	43	INFIL.BAC	79	CROSSQ.OUT
10	CADPTS.DAT	44	INFLOW.BAC	80	HYCHAN.OUT
11	SUMMARY.OUT	45		81	FINALVEL.OUT
12	Used	46	CHAN.BAC	82	used
13	Used	47	ARF.BAC	83	FINALVELDIR.OUT
14	CHANMAX.OUT	48	MULT.BAC	85	XSEC.DAT
15	Used	49	SED.BAC	86	CHANBANKEL.CHK
16	Used	50	OUTFLOW.DAT	87	OVERBANK.OUT
17	SEDFP.OUT	51	OUTFLOW.BAC	88	SEDTRAN.OUT
18	SEDCHAN.OUT	52	STREET.DAT	89	RAINCELL.DAT
19	TIME.OUT	53	STREET.BAC	90	CHANSTABILITY.OUT
20	DEPTH.OUT	54	CROSSMAX.OUT	91	XSECAREA.OUT
21	TIMDEP.OUT	55	SURFAREA.OUT	92	INTERGWS.OUT
22	ROUGH.OUT	56	CHANNEL.CHK	93	CHANWS.OUT
23	STREET.OUT	57	LEVEE.DAT	94	XSEC.OUT
24	VELOC.OUT	58	LEVEE.BAC	95	EVAPOR.DAT
25	MULTCHN.OUT	59	LEVEE.OUT	96	HYCROSS.OUT
26	FPREV.NEW	60	DEPFP.OUT	97	TOPO.ORI
27	DEPTHTOL.OUT	61	DEPCH.OUT	98	used
28	DEPTHDUR.OUT	62	FINALDEP.OUT	99	used
29	TOLER.BAC	63	VELDIREC.OUT	100	
30	CONT.DAT	64	VELFP.OUT	101	IMPACT.OUT
31	FPLAIN.DAT	65	SUPER.OUT	102	STATICPRESS.OUT
32	RAIN.DAT	66	CHVOLUME.OUT	103	CONFLUENCE.OUT
33	INFIL.DAT	67	INFILHY.OUT	104	SPECENERGY.OUT
34	INFLOW.DAT	68	HYDROSTRUC.DAT		
35	EVAPOR.BAC	69	HYDROSTRUC.BAC		
36	CHAN.DAT	70	HYDROSTRUCT.OUT		
37	ARF.DAT	71	used		
38	MULT.DAT	72	STVELDIR.OUT		
39	SED.DAT	73	STVEL.OUT		

4.3 Data Files

The following data files have been organized by components. The first five data files are required for every flood simulation. The CADPTS.DAT is not listed in the following the data files and it is required for a flood simulation, however, this file is complete and will not need further editing unless grid elements are added at a later time. It contains the grid element, x-coordinate and y-coordinate for creation of the output files. Each required FLO-2D data input file is identified by a tab for easy access.

FILE: CONT.DAT
SYSTEM CONTROL VARIABLES

Line 1 contains simulation time, output interval, display control, units switch and back-up file.

1 **SIMUL, TOUT, LGPLOT, METRIC, IBACKUP**

Line 2 contains system component switches.

2 **ICHANNEL, MSTREET, LEVEE, IWRFS, IMULTC**

Line 3 is a list of physical processes option switches.

3 **IRAIN, INFIL, IEVAP, MUD, ISED**

Line 4 has conveyance structure, floodway and debris basin option switches.

4 **IHYDRSTRUCT, IFLOODWAY, IDEBRV**

Line 5 contains parameters for global alteration of the data files.

5 **AMANN, XCONC, XARF, FROUDL, SHALLOWN, ENCROACH**

**** NOTE: IF IFLOODWAY IS EQUAL TO ZERO, OMIT ENCROACH ****

Line 6 is the print option for the floodplain and the supercritical flow file switch.

6 **NOPRTFP, SUPER**

**** NOTE: IF ICHANNEL IS EQUAL TO 0, OMIT LINE 7 ****

Line 7 is the print option for the channels.

7 **NOPRTC**

Line 8 is the graphics display time interval update.

**** NOTE: IF LGPLOT IS EQUAL TO ZERO, OMIT LINE 8 ****

8 **GRAPTIM, INPLOT**

An example of the CONT.DAT file:

```
100.0 0.100 2 0 0
1 1 0 0 0
0 1 0 0 0
0 0 0
0.000 0.0 0.30 0.70 0.150
2 0
2
0.1 1
```

VARIABLE DESCRIPTIONS FOR THE CONT.DAT FILE

(i) = Integer variable (r) = Real variable

Variable	Description
AMANN -	(r) Incremental change to the floodplain Manning's n roughness coefficient. A positive value for AMANN will uniformly increase the n-value for every floodplain grid element in the system. If the n-value is less than or equal to 0.03, that roughness coefficient will not be increased for that grid element.
ENCROACH –	(r) The floodway encroachment increase in flow depth (ft or meters). 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) Maximum Froude number for overland flow. When FROUDL is exceeded, the floodplain n-value is increased by 0.001 for that grid element for the next timestep (see comment 3). Set FROUDL = 0.0 for no Froude number limitations.
GRAPTIM -	(r) Time interval in hours that the graphics display is updated (e.g. set GRAPTIM = 0.02 for a frequent update). GRAPHTIM is required when LGPLOT = 2. This variable will not affect the file output data time interval (TOUT). The graphics mode is limited to a 48-day hydrograph.
IBACKUP -	(i) Switch, 0 or 1; IBACKUP = 1 writes input data to backup file with the file name plus *.BAC extension.
ICHANNEL -	(i) Switch, 0 or 1; If ICHANNEL = 1 main channel flow will be simulated. The CHAN.DAT must be created (see comments 1 and 6).
IDEBRV -	(i) Switch, 0 or 1; Set IDEBRV = 1 if a debris basin volume should be filled before routing the flow hydrograph.
IEVAP -	(i) Switch, 0 or 1; if simulating free water surface evaporation from overland flow or channel flow, set IEVAP = 1.
IFLOODWAY –	(i) Switch, 0 or 1; If FLOODWAY =1 a floodway analysis will be performed in the next FLO-2D simulation. A FLO-2D flood simulation must be completed to determine the maximum floodplain water surface elevations prior to initiating a floodway simulation (see comment 5).
IHYDRSTRUCT -	(i) Switch, 0 or 1; Setting IHYDRSTRUCT = 1 indicates that one or more hydraulic structures either on the floodplain or in the channel will be simulated. The HYSTRUC.DAT file must be created.

VARIABLE DESCRIPTIONS FOR THE CONT.DAT FILE (cont.)

(i) = Integer variable (r) = Real variable

Variable	Description
IMULTC -	(i) Switch, 0 or 1; Set IMULTC = 1 to simulate multiple channel (rill and gully flow) on the floodplain. Rill and gully flow rather than overland sheet flow will be simulated to route the flow between designated floodplain grid elements. The MULT.DAT file must be created.
INFIL -	(i) Switch, 0 or 1; 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.
INPLOT -	(i) Switch, 0, 1 or 2; INPLOT controls inflow hydrograph plotted on the graphics screen. LGPLOT must be 2 to plot the inflow hydrograph (see comment 4). INPLOT = 0, to plot the floodplain inflow hydrograph with or without rainfall INPLOT = 1, to plot the channel inflow hydrograph INPLOT = 2, to plot the rainfall without an inflow hydrograph.
IRAIN -	(i) Switch 0 or 1; Set IRAIN = 1 to simulate rain on the grid system. The RAIN.DAT file must be created (see comment 1).
ISED -	(i) Switch 0 or 1; If ISED = 1, the sediment transport routine will be used. The SED.DAT file must be created.
IWRFS -	(i) Switch 0 or 1; IWRFS = 1 specifies that area and width reduction factors (ARFs and WRFs) will be assigned in the ARF.DAT file.
LEVEE -	(i) Switch 0 or 1; Set LEVEE = 1 to simulate levees. The LEVEE.DAT must be created.
LGPLOT -	(i) Switch 0 or 2; LGPLOT = 0 will display a text message on the screen which scrolls the simulation time, minimum timestep and volume conservation. LGPLOT = 2 provides a graphic display of the floodwave progression over the grid system (flow depth) and inflow hydrograph.
METRIC -	(i) Switch, 0 or 1; METRIC = 0 for English units and METRIC = 1 for the metric system of units.
MSTREET-	(i) Switch, 0 or 1; Assign MSTREET = 1 to initiate the street flow component. The STREET.DAT file must be created (see comment 2).

VARIABLE DESCRIPTIONS FOR THE CONT.DAT FILE (cont.)

(i) = Integer variable (r) = Real variable

Variable	Description
MUD -	(i) Switch, 0 or 1; Set MUD = 0 for clear water and MUD = 1 for hyperconcentrated sediment flow. Setting MUD = 1 requires a sediment concentration by volume. Either the variable HP(I,J,3) for a floodplain hydrograph or H(I,J,3) for a channel hydrograph to be assigned to each inflow hydrograph interval (see comments 1 and 3). The SED.DAT file must be created.
NOPRTC -	(i) Switch, 0, 1, or 2; If NOPRTC = 0, all the channel data is printed to the BASE.OUT file; If NOPRTC = 1, the channel outflow data is not printed to the BASE.OUT file; If NOPRTC = 2, all the channel output data is not printed.
NOPRTFP -	(i) Switch, 0, 1, 2 or 3; If NOPRTFP = 0, all the floodplain flow data is printed to the BASE.OUT file; If NOPRTFP = 1, floodplain outflow data is not printed to the BASE.OUT file; If NOPRTFP = 2, the floodplain flow data is not printed to the BASE.OUT file; If NOPRTFP = 3, only floodplain outflow data is printed to the BASE.OUT file.
SHALLOWN -	(r) Flow roughness n-value for shallow overland flow (flow depth < 0.2 ft or 0.6 m). When flow depths greater than 0.2 ft but less than 0.5 ft, the shallow overland flow n-value is decreased by 50%. If 0.2 ft < flow depth < 0.5 ft, then SHALLOWN = SHALLOWN/2. See Table 3, page 26 of the Users Manual.
SIMUL -	(r) Simulation time (hours).
SUPER -	(r) Switch 0 or 1. SUPER = 1 prints a message that the flow is supercritical in the given grid element to the SUPER.OUT file. IF SUPER = 0, the message will be not be printed for either overland and channel flow.
TOUT -	(r) Output time interval (hours). This is time interval that hydraulic data is recorded in the various output files *.OUT.
XARF -	(r) Global area reduction factor applied to all grid elements. This factor reduces the grid element surface area available for flood volume storage. XARF can be used to account irregular surface topography, dense vegetation or other features. Range: 0 < XARF < 1. A typical value for XARF might be 0.10 indicating that 10% of each grid element surface is not available for flood storage. The XARF value is overridden by the ARF variables specified for the individual grid elements in the ARF.DAT file. Assign XARF = 0 to flood the entire surface area of the grid elements.

VARIABLE DESCRIPTIONS FOR THE CONT.DAT FILE (cont.)

(i) = Integer variable (r) = Real variable

Variable	Description
XCONC -	(r) 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 for the CONT.DAT File

The following comments will assist in the developing of the CONT.DAT file:

1. If any of the switches MUD, ISED, IRAIN, IMULT, INFIL, MSTREET, LEVEE, ICHANNEL, IWRFS or IHYDRSTRUCT are equal to 0, then the corresponding data file and all other related data can be omitted. For example, for an overland flow model without a streets, set MSTREET = 0 and the STREET.DAT file can be omitted.

2. Streets, mudflow, levees, and rill and gully flow can be simulated with or without a channel.

3. On alluvial surfaces, supercritical flow is uncommon and may be inhibited by sediment entrainment. To avoid supercritical overland flow or any high Froude number flow, set FROUDL to a maximum anticipated Froude number. There are three approaches to supercritical or high Froude number flow analysis:

   a. Allow supercritical flow and do not limit the Froude number.
   b. Increase the roughness for the entire flow domain by increasing AMANN or set higher individual grid element n-values to reduce the Froude number.
   c. Limit the Froude number to a specific value (e.g. set FROUDL = 0.99 or 1.11, etc.) When FROUDL is exceeded the grid element roughness values will be increased by 0.001 for the next timestep. The flow Froude number will only momentarily exceed FROUDL. After completion of a flood simulation, review the ROUGH.OUT file to determine where FROUDL was exceeded and the maximum n-values that were computed. You may consider revising the n-values in the FPLAIN.DAT file to match those in the ROUGH.OUT file if you want to ensure that FROUDL is not exceeded.

   You should review the floodwave travel time to determine if it is appropriate. Conversely, use the travel time to calibrate the n-values. Adjusting n-values with FROUDL or the other Froude number variables (FROUDC for channel flow and STFNO for street flow) will slow the arrival of the frontal wave. During the falling limb of the hydrograph, when the Froude number decreases to less than 0.5 and the flow becomes shallow the n-value decreases by 0.0005 until the originally specified n-value is reached.

4. The INPLOT variable determines whether a floodplain or channel inflow hydrograph is displayed. If rainfall is being simulated, the cumulative total precipitation will be plotted.

5. IFLOODWAY initiates the floodway routine. Flow will not be exchange between floodplain grid elements unless the maximum water surface plus the encroachment depth (ENCROACH) from a previous FLO-2D simulation is exceeded. See the Floodway discussion on page 36 in the Manual component descriptions.

6. If channel flow is simulated and ICHANNEL = 1, then the NOPRTC variable must be set in CONT.DAT and the DEPTOLCH variable must be defined in TOLER.DAT. If the channel has an inflow hydrograph, it must be created in INFLOW.DAT file. In addition, any channel outflow control must be defined in OUTFLOW.DAT.

FILE: TOLER.DAT
CONTROL VARIABLES

Line 1 contains stability criteria and tolerance variables.

**** NOTE: If channel flow is simulated use either Line 1a or 1b ****
**** If KMODEL is 3 (full dynamic wave), use either Line 1a or 1c ****

1a **KMODEL, TOL, DEPTOLFP, DEPTOLCH, WAVECEL, WAVEMIN, WAVEMAX**

1b **KMODEL, TOL, DEPTOLFP, DEPTOLCH**

1c **KMODEL, TOL, DEPTOLFP, WAVECEL, WAVEMIN, WAVEMAX**

1d **KMODEL, TOL, DEPTOLFP**

Line 2 contains MAXPLOT or MAPPER animation variables.

2 **ITIMTEP, TIMTEP**

Line 3 contains limiting timestep variables

3 **DTMIN, DTMAX**

An example of the TOLER.DAT file:

```
3   0.1   0.30   0.25   0.20   1.0   4.0
0   0
0.10   60.
```

VARIABLE DESCRIPTIONS FOR THE TOLER.DAT FILE

(i) = Integer variable (r) = Real variable

Variable **Description**

DEPTOLFP - (r) Tolerance value for the percent change in floodplain flow depth for a given timestep. If DEPTOLFP is exceeded for any grid element, the timestep will be reduced. If DEPTOLFP = 0, then the timestep is governed by the prescribed numerical stability criteria (see comment 3). Suggested value for DEPTOLFP = 0.3. Range: 0.1 to 0.5.

DEPTOLCH - (r) Tolerance value for the percent change in channel flow depth for a given timestep. Suggested values of DEPTOLCH are 0.1 to 0.5 (see comment 3).

DTMIN - (r) Minimum timestep (seconds). This variable should only be decreased if ther are numerous messages of exceeding the stability criteria (see comment 4). Suggested value = 1.0 second.

DTMAX - (r) Maximum timestep (seconds). Suggested value = 30 secs (see comment 4).

ITIMTEP - (i) Switch, 0 or 1; set ITIMTEP = 1 for to create file of time lapse flow depths for a MAXPLOT or MAPPER post-simulation flood animation. The timestep TIMTEP must be specified.

KMODEL - (i) Switch, 2 or 3; KMODEL = 2 prescribes the diffusion wave equation for flood routing. If KMODEL = 3 the full dynamic wave equation is used for both the channel and floodplain (see comment 1).

TIMTEP - (r) An output time interval (hrs) that the flow depth will be written to an output file for a post-simulation flood animation in MAXPLOT or MAPPER. TIMTEP should be a multiple of TOUT. The variable ITIMTEP must be set.

TOL - (r) Surface detention (feet or meters). TOL is a minimum value of the flow depth for flood routing. A typical value is 0.10 ft (see comment 2). Use a small TOL value for rainfall runoff (0.025 ft to 0.05 ft).

WAVECEL - (r) A wave celerity weighting factor for dynamic routing ($0 < \zeta_w < 10$). WAVECEL controls the rate of change in the wave celerity ζ assigned to each grid element (see pg. 18 User's Manual). WAVECEL controls how fast the wave celerity ζ is incremented and decremented. Initially set WAVECEL 0.1 times WAVEMIN. Review comment 5 and the following WAVEMIN and WAVEMAX descriptions.

VARIABLE DESCRIPTIONS FOR THE TOLER.DAT FILE (cont.)

(i) = Integer variable (r) = Real variable

Variable **Description**

WAVEMAX - (r) Maximum value of the numerical stability coefficient ζ for full dynamic wave flood routing. Initially set $\zeta_{max} = \sim 10 \times \zeta_{min}$. WAVEMAX directly controls the maximum timestep for the full dynamic wave. Reduce WAVEMAX to improve volume conservation or reduce surging. Increase WAVEMAX to speed up the flood simulation.

WAVEMIN - (r) Minimum value of the numerical stability coefficient ζ for full dynamic wave flood routing. ζ_{min} will generally range from $0.1 < \zeta_{min} < 10$. Initially set $\zeta = 1.0$ to 5.0. WAVEMIN limits the minimum timestep. Reduce WAVEMIN to improve volume conservation. Increase WAVEMIN to speed up the flood model.

Instructional Comments for the TOLER.DAT File

1. The flood routing can be simulated as either a diffusive or dynamic wave using the KMODEL switch (KMODEL = 2 for diffusive wave and KMODEL = 3 for dynamic wave). Channel and overland flow routing are simulated with the same momentum equation.

2. The TOL value prescribes the flow depth for an individual grid element below which no flood routing will be performed. TOL is analogous to a depression storage abstraction value.

3. DEPTOLFP and DEPTOLCH control the percent change in flow depth for a given timestep in the routing sequence. They affect the computer runtime and flow depth resolution. If a flood simulation yields poor results either by failing to conserve volume or by noted erratic or surging discharge, then these parameters too low for numerical stability. Decreasing either of these two values in a range from 0.05 to 0.20 will reduce the timestep and improve the numerical stability. Lower values will also result in more flow depth iterations, higher resolution, and longer computational times. Setting DEPTOLFP equal to 0 dictates that only the Courant stability criteria will be applied for numerical stability. Significant numerical instability may require revisions in channel slope, flow area or roughness. Erratic discharge may be a related to mudflow surging or channel return flow and may not always indicate numerical instability.

4. The suggested minimum and maximum timesteps have been found to be appropriate for virtually all flood simulations. These values may have to be adjusted for the following cases:

   i. River channel flooding with seasonal daily (instead of hourly) hydrographs. Both DTMIN and DTMAX should be increased in this case.

   ii. Flood simulations with small grid elements (less than 50 ft on a side). DTMIN may have to be less than 1 second. Decrease DTMIN to 0.1 or 0.01 seconds for difficult simulations. If DTMIN must be less than 0.01 to maintain numerical stability, the grid elements may be too small.

5. Decreasing the wave celerity ζ decreases the timestep and improves the numerical stability and volume conservation for the full dynamic wave. WAVECEL controls the rate of increase of the wave celerity from between the lower (WAVEMIN) and upper (WAVEMAX) limits. By increasing WAVEMIN and WAVEMAX, the flood simulation speeds up by using larger timesteps but also permits greater variation in the water surface elevations that can adversely affect the numerical stability and volume conservation. Review the CHANMAX.OUT file, the HYDROG program hydrograph plots and the BASE.OUT file for hydrograph spikes and dips (surging) or for failure to conserve volume. If surging or volume conservation problems are observed, reduce WAVEMAX first. To speed up stable simulations, experiment with higher values of WAVEMAX and WAVEMIN. For mild channel slopes, WAVEMIN should be set to 2 or lower and WAVEMAX should be less than 5. The TIME.OUT file will identify which floodplain or channel grid elements cause the most timestep decreases.

FILE: FPLAIN.DAT
FLOODPLAIN ELEMENT VARIABLES

Line 1 is the floodplain grid elements, Manning's n roughness value and grid elevation.

1 **DUM, (FP(I, J); I = 1, NNOD; J = 1, 6)**

A partial example of the FPLAIN.DAT file:

```
 1     0     2     0     0 0.065   4680.77
 2     3     4     0     1 0.065   4760.07
 3     0     5     2     0 0.065   4671.49
 4     5     7     0     2 0.065   4904.23
 5     6     8     4     3 0.065   4734.29
 6     0     9     5     0 0.065   4673.27
 7     8    11     0     4 0.065   4921.95
 8     9    12     7     5 0.065   4803.38
 9    10    13     8     6 0.065   4696.37
10     0    14     9     0 0.065   4672.92
11    12    26     0     7 0.065   4913.47
12    13    27    11     8 0.065   4812.28
13    14    28    12     9 0.065   4695.81
14    15    29    13    10 0.065   4677.02
15    16    30    14     0 0.065   4686.69
16    17    31    15     0 0.065   4757.84
17    18    32    16     0 0.065   4885.13
18    19    33    17     0 0.065   4910.89
19     0    34    18     0 0.065   5072.85
20    21    35     0     0 0.065   4952.39
21    22    36    20     0 0.065   4881.80
22    23    37    21     0 0.065   4844.74
23    24    38    22     0 0.065   4862.00
24    25    39    23     0 0.065   4918.34
25    26    40    24     0 0.065   4946.60
26    27    41    25    11 0.065   4934.34
27    28    42    26    12 0.065   4748.42
28    29    43    27    13 0.065   4662.20
29    30    44    28    14 0.065   4695.74
30    31    45    29    15 0.065   4699.40
31    32    46    30    16 0.065   4887.07
32    33    47    31    17 0.065   4954.84
33    34    48    32    18 0.065   4987.67
34     0     0    33    19 0.065   4994.18
35    36    52     0    20 0.065   4921.99
36    37    53    35    21 0.065   4883.19
37    38    54    36    22 0.065   4879.23
38    39    55    37    23 0.065   4788.36
```

VARIABLE DESCRIPTIONS FOR THE FPLAIN.DAT FILE

(i) = Integer number (r) = Real number

Variable	Description
DUM -	(i) 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) Floodplain element contiguous to grid element I 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) Average Manning's n roughness coefficient assigned to grid element I.
FP(I,6) -	(r) Average ground-surface elevation for grid element I (ft).

Instructional Comments for the FPLAIN.DAT File

This file is usually prepared by the grid developer system program GDS and generally require no data revisions. Occasionally, an n-value or elevation for an individual grid element is modified. Spatial variability and editing of both the n-values and elevations are easily accomplished in the GDS or FLOENVIR programs. If grid elements are added to the FPLAIN.DAT file after creating the file, there are several rules that should be observed.

1. It is important to review the grid system before proceeding with the flood simulation. Check the FPLAIN.DAT file with the GDS grid system image to ensure that the contiguous elements are properly correlated. It is suggested to check a few grid elements at each end of the grid system

2. 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.

3. The elements should be numbered consecutively starting with 1.

4. 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.

5. Any additional grid elements in the FPLAIN.DAT file must have corresponding grid elements in the CADPTS.DAT file.

FILE: INFLOW.DAT
INFLOW HYDROGRAPH VARIABLES

Line 1 contains the number of inflow hydrographs, the inflow time interval switch, and the inflow grid element to be plotted.

**** NOTE: IF ONLY RAINFALL IS BEING SIMULATED, SET NOHYDRO = 0 ****
**** AND OMIT THE REST OF LINE 1 ****

1 **NOHYDRO, IHOURDAILY, IDEPLT**

**** NOTE: IF NOHYDRO EQUALS 0, OMIT LINES 2 AND 3 ****
**** REPEAT LINES 2 AND 3 FOR EACH INFLOW GRID ELEMENT ****

Line 2 is the inflow character, inflow or diversion switch and the grid element number.

2 **IFC(I) = 'F' or 'C', INOUTFC(I), KHIN(I); I = 1, NOHYDRO**

Line 3 is the hydrograph character and discretized hydrograph entries (time, discharge, sediment concentration by volume or sediment volume).

3 **HYDCHAR = 'H', HP(I,J,1), HP(I,J,2), HP(I,J,3); I = 1, NOHYDRO; J = 1, Number of inflow hydrograph pairs**

**** NOTE: IF MUD = 0,* ***HP(I,J,3)*** *IS OMITTED. ****

An example of the INFLOW.DAT file:

```
5    0   4335
C            0    4335
H            0       0
H            1    5530
H            2    5530
H            3    5378
H            4    5378
H            5    5227
H            6    5075
H            7    4924
H            8    4772
H            9    4772
H           10    4621
H           11    4469
H           12    4469
H           13    4469
H           14    4469
H           15    4469
H           16    4469
```

VARIABLE DESCRIPTIONS FOR THE INFLOW.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

HP(I,J,1) - (r) Time corresponding to the start of the floodplain inflow hydrograph interval (hours). The first hydrograph time-discharge set should be 0.0 and 0.0.

HP(I,J,2) - (r) Floodplain discharge (cfs) corresponding to the time interval which starts at HP(I,J,1).

HP(I,J,3) - (r) Sediment concentration by volume when HP(I,J,3) is less than 1.0 ($0. \leq HP(I,J,3) \leq 1.0$). HP (I,J,3) corresponds to the 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 (see comment 2).

HYDCHAR - (c) Character 'H' that identifies Line 3 inflow hydrograph time and discharge pairs. Each line of the hydrograph begins with 'H'.

IDEPLT - (i) 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.

IFC(I) - (c) Character 'F' or 'C' to identify if the inflow hydrograph grid element is a floodplain 'F' or a channel 'C'. See comment 1.

INOUTFC(I) - (i) Switch 0 or 1; INOUT = 0 for inflow hydrograph. INOUT = 1 for outflow hydrograph or diversion from the channel.

IHOURDAILY - (i) Switch, 0 or 1; Set IHOURDAILY = 0 for hourly intervals in the discretized inflow hydrograph HP(I,J,1). Set IHOURDAILY = 1 for daily (24 hr) intervals of HP(I,J,1).

KHIN(I) - (i) Array of grid elements with a inflow hydrograph (inflow nodes).

NOHYDRO – (i) Number of inflow hydrograph grid elements for either the floodplain or the channel.

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 an inflow sediment volume for the time interval starting at HP(I,J,1). The sediment volume is input in ft^3 or m^3 and can represent erosion, hillslope failure, or any other type of mass sediment loading. For mudflow sediment concentrations 20% by volume or less, the flow is routed as bulked water flows.

FILE: OUTFLOW.DAT
OUTPUT VARIABLES

Line 1 is number of floodplain outflow grid elements.

1 **NDC**

**** IF NDC = 0, OMIT LINE 2 ****

Line 2 contains the floodplain outflow grid elements, separated by spaces.

2 **NODDC(I), I = 1, NDC**

****IF ICHANNEL = 0 IN CONT.DAT, OMIT LINES 3, 4 and 5****

Line 3 is the number of channel outflow elements.

3 **NCHO**

**** IF NCHO IS EQUAL TO ZERO, OMIT LINES 4 and 5 ****

Line 4 is the channel outflow grid element and number of outflow stage-discharge pairs.

4 **KOUT(N), NPCHO (N); N = 1, NCHO**

**** IF NPCHO IS EQUAL TO ZERO, OMIT LINE 5 ****

Line 5 is the array of stage-discharge relationship as assigned by depth.

5 **HOUT(N,J,1), HOUT(N,J,2), HOUT(N,J,3); N=1, NCHO; J=1, NPCHO(N)**

Line 6 is number of floodplain cross sections and print control for cross section output.

6 **NXSEC**

**** IF NXSEC IS EQUAL TO 0, OMIT LINE 7 and 8 ****

Line 7 is the print control for the cross section summary data.

7 **NXPRT**

**** LINE 8 IS REPEATED NXSEC TIMES ****

Line 8 is an array of cross section flow directions and grid elements.

8 **IFLO(N), NNXSEC(N), (NODX(N,J); N = 1, NXSEC; J = 1, NNXSEC(N)**

FILE: OUTFLOW.DAT (cont.)
OUTPUT VARIABLES

Line 9 is the number of floodplain or channel elements with stage-time relationships.

9 **NSTA**

**** IF NSTA IS EQUAL TO 0, OMIT LINES 10 and 11 ****
**** REPEAT LINES 10 and 11 FOR EACH ELEMENT ****
**** WITH A TIME-STAGE RELATIONSHIP ****

Line 10 is the number of grid elements, channel or floodplain identifier and the number of stage-time relationships.

10 **NOSTA(N), NOSTACFP(N), NPSTA(N); N = 1, NSTA**

Line 11 is the stage-time relationship for each element.

11 **STA_TIME(N,J), STA_STAGE(N,J); N = 1, NSTA, J = 1, NPSTA(N)**

Line 12 is the flow depth for a flood duration analysis.

12. **DEPTHDUR**

An example of the OUTFLOW.DAT file follows:

```
 94
  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      37      38      39      40
 41      42      43      44      45      46      47      48      49      50
 51      52      53      54      55      56      57      58      59      60
 61      62      63      64      65      66      67      68      69      70
 71      72      73      74      75      76      77      78      79      80
 81      82     164     245     326     407     488     569     650     731
812     893     974    1055
  1
 48      0
  0
  0
0.5
```

VARIABLE DESCRIPTIONS FOR THE OUTFLOW.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

DEPTHDUR - (r) 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 9).

HOUT(I,J,1) - (r) Array of channel maximum depths for which a channel outflow stage-discharge relationship is valid.

HOUT(I,J,2) - (r) Array of coefficients for the channel element (I) outflow stage-discharge relationship (see comment 3).

HOUT(I,J,3) - (r) Array of exponents for the channel element (I) outflow stage-discharge relationships.

IFLO(N) - (i) Defines the general direction that the flow is expected to cross the floodplain cross section. If the general flow direction is to the south (coming from the north), set IFLO = 3, where 3 = south direction. There are eight possible directions. 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. For example IFLO = -1 eliminates the flow components from the 5 and 8 directions. See comment 5.

KOUT(I) - (i) Array of channel outflow elements. These elements discharge flow out of the grid system from the channel.

NCHO - (i) Number of channel outflow elements.

NDC - (i) Number of floodplain outflow elements.

NNXSEC(N) - (i) Number of floodplain elements in a given cross section. The number of grid elements does not have to constitute an entire cross section (e.g., only one grid element out of many on the floodplain may be specified).

NODDC(I) - (i) Array of floodplain outflow grid elements. These elements discharge flow out of the grid system from the floodplain (see comment 1).

NODX(N,J) - (i) Array of grid elements that constitute a given floodplain cross section (see comment 6).

NOSTA(I) - (i) Array of channel elements with stage-time relationships. If NOSTA(I) is a inflow grid element, assign NOSTA(I) as a negative value to compute inflow volume (see comment 8).

VARIABLE DESCRIPTIONS FOR THE OUTFLOW.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable **Description**

NOSTACFP(I) – (i) 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.

NPCHO (N) - (i) Number of channel outflow hydrograph pairs. If NPCHO = 0, the sum of all inflow to this outflow element from the surrounding channel elements will be discharged from the grid system (see comment 2).

NPSTA (N) - (i) Number of pairs of channel stage-time entries (see comment 7).

NSTA - (i) Number of channel stage-time elements.

NXPRT - (i) Switch 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.

NXSEC - (i) The number of floodplain cross sections to be analyzed. If NXSEC greater than zero then those elements in the specified cross sections will be grouped together in the output data (see comment 4).

STA_TIME(N,J) - (r) Array of time intervals (hrs) for the grid element stage-time relationship.

STA_STAGE(N,J,) - (r) Array of water surface elevations (ft or m) for the stage-time relationship.

Instructional Comments for the OUTFLOW.DAT File

1. Either the channel or the floodplain outflow elements can be used to discharge the flow from the grid system. The outflow node is an artificial grid element whose sole purpose is to discharge flow off the grid system. The outflow nodes should not contain hydraulic structures, streets or other attributes. The floodplain elevation of the outflow node is automatically set to an elevation lower (0.25 ft or 0.1 m) than the lowest upstream grid element unless it is already lower than all the upstream grid elements.

2. Setting NPCHO = 0 will discharge all the inflow to this channel outflow element from the grid system. This outflow discharge is determined by setting the outflow equal to the sum of the inflow from the neighboring channel elements. This method approaches normal flow depth in the outflow elements. This is a simple method to ensure that backwater related to arbitrary boundary conditions does not occur in the upstream elements.

3. Channel boundary outflow condition may be established by specifying a stage-discharge relationship given by $Q = a\ h^b$ where the coefficient (a) and exponent (b) are required input and h is the flow depth. The coefficient (a) and exponent (b) can be used to established critical flow at the outflow grids.

4. If NXSEC = 0, then no floodplain cross sections will be analyzed and the cross section grid elements should be excluded from the data file. The floodplain grid elements can be combined to define a cross section across the floodplain or alluvial fan. Setting NXSEC = I initiates this option. The cross section grid elements must be specified.

5. Floodplain cross section analyses are permitted 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 components are as follows:

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

6. If a grid element is listed in more than one cross section, then the individual grid element hydrographs in the output file CROSSQ.OUT will not be correct. Output data in the CROSSMAX.OUT and HYCROSS.OUT files, however, will be correct.

Instructional Comments for the OUTFLOW.DAT File (cont.)

7. A discretized time-stage relationship can be employed to specify a water surface elevation for one or more grid elements. This relationship can be used to specify flow conditions at various channel locations in the grid system. This is a simple method by which to simulate storm surge flooding on the coastal floodplain. The outflow elements can all be specified with increasing tides or surge water surface elevations.

8. If a water surface elevation is specified with NSTA, check to see if one of the channel elements specified by NOSTA(I) is an inflow channel element. If NOSTA(I) is an inflow element in the CINOUT.DAT file, set NOSTA(I) as negative value to compute the inflow volume at this channel element which corresponds to the constant water surface elevation.

9. The depth duration analysis is used to determine how long a floodplain grid element is inundated at a flow depth greater than the specified 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 are plotted in MAXPLOT.

FILE: RAIN.DAT
RAINFALL VARIABLES

**** NOTE: IF IRAIN = 0 IN CONT.DAT OMIT THIS FILE ****

Line 1 is a real time rainfall data switch.

1 **IRAINREAL**

Line 2 is the total rainfall, rainfall abstract, switch for using spatially variable depth-area reduction values and the moving storm switch.

2 **RTT, RAINABS, IRAINARF, MOVINGSTORM**

Line 3 includes a rainfall character, rainfall time and distribution.

3 **RAINCHAR = 'R', R_TIME(I); R_DISTR(I); I = 1, number of rainfall pairs.**

**** NOTE: IF MOVINGSTORM = 0, OMIT LINE 4 ****

Line 4 is the moving storm speed and direction.

4 **RAINSPEED, IRAINDIR**

**** NOTE: IF IRAINARF = 0, OMIT LINE 5 ****

Line 5 has depth-area reduction spatial variation listing grid element and depth-area reduction.

5 **IRGRID(I), RAINARF(I); I = 1, number of rainfall depth-area reduction values.**

An example of the RAIN.DAT file:

```
0
     3.100          0.000     0       0
 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   0.417          0.500
 R   0.500          0.550
 R   0.583          0.600
 R   0.667          0.680
 R   0.750          0.750
 R   0.833          0.780
 R   0.917          0.840
 R   1.000          0.860
 R   1.083          0.880
 R   1.167          0.900
 R   1.250          0.910
 R   1.330          0.930
 R   1.420          0.950
 R   1.500          0.970
 R   1.580          0.980
 R   1.670          0.985
 R   1.750          0.990
 R   1.830          0.995
 R   1.920          0.998
 R   2.000          1.000
```

VARIABLE DESCRIPTIONS FOR THE RAIN.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

IRAINARF - (i) Switch; 0 or 1; IRAINARF = 1 indicates that individual grid element depth-area reduction values will be assigned.

IRAINDIR - (i) Direction of the moving storm (one of the eight potential flow directions where 1 = north, 2 = east, 3 = south, 4 = west, 5 = NE, 6 = SE, 7 = SW, 8 = NW).

IRAINREAL - (r) Switch; 0 or 1; IRAINREAL = 1 indicates that real-time rainfall will be simulated. The RAINCELL.DAT file containing the spatial and temporal rainfall data must be prepared by the GDS.

IRGRID - (i) Grid element with a spatially defined rainfall depth area reduction value. This data is automatically generated in the FLOENVIR processor program

MOVINGSTORM – (i) Switch; 0 or 1; where 1 indicates that a moving storm will be simulated.

RAINABS - (r) Rainfall interception and abstraction (inches or mm) if infiltration is not being modeled (see comment 2).

RAINARF - (r) Rainfall depth area reduction to create spatially variable rainfall. This data is automatically generated in the FLOENVIR processor program.

RAINCHAR – (a) Character 'R' that identifies Line 3.

RAINSPEED – (r) Storm speed (mph or mps).

RTT - (r) Total storm rainfall (inches or mm).

R_TIME(I) - (r) Time (hours) corresponding to the start of the specified rainfall interval.

R_DISTR(I) - (r) Rainfall intensity distribution as a cumulative percentage of the total storm which initiates at the time interval R_TIME(I). R_DISTR(I) varies as (0. < R_DISTR(I) < 1.). See comment 1.

Instructional Comments for the RAIN.DAT File

1. The rainfall distribution has to be related to the overall flood simulation time. The rainfall may occur for only a portion of the total flood simulation and may start later than the flood simulation time represents. For most rain storms, the start of the simulation correlates with the rainfall distribution. In those cases, where the rainfall and the simulation time are not correlation, it may be necessary to use 0.0 cumulative rainfall at the beginning of the flood simulation for a period of time and similarly 1.0 (100%) cumulative rainfall near the end of the simulation.

2. If infiltration is being simulated, set the RAINABS = 0 and assign the rainfall abstraction in the INFIL.DAT file.

FILE: INFIL.DAT
INFILTRATION VARIABLES

**** NOTE: IF INFIL = 0 IN THE CONT.DAT FILE, OMIT THIS FILE ****

Line 1 contains the abstraction, initial and final saturation, soil porosity and the switch for channel infiltration.

1 **ABSTR, SATI, SATF, POROS, INFCHAN**

Line 2 is the global assignment of hydraulic conductivity and soil suction parameters to the entire grid system.

2 **HYDCALL, SOILALL**

Line 3 is the floodplain hydraulic conductivity and soil suction for individual grid elements that supersede the values in line 2.

3 **INFILCHAR = 'F', INFGRID(N), HYDC(N), SOILS(N), DTHETA(N), ABSTRINF(N), RTIMPF(N); N = 1, end of individual floodplain element values**

**** NOTE: IF INFCHAN = 0, IGNORE THE REST OF THE FILE ****

Line 4 is the channel hydraulic conductivity for all the channel elements.

4 **HYDCHN**

Line 5 is the channel hydraulic conductivity for individual grid elements that supersede the values in Line 4.

5 **INFILCHAR = 'C', INFCH(N), HYDCONCH(N); N = 1, end of the individual channel hydraulic conductivity values**

An example of the INFIL.DAT file for floodplain infiltration only:

```
0.000    0.90    1.00   0.40   0
0.02     6.80
F    1588     0.04     5.40
F    1589     0.04     5.40
F    1590     0.04     5.40
F    1591     0.04     5.40
F    1592     0.04     5.40
F    1593     0.04     5.40
F    1594     0.04     5.40
F    1653     0.05     5.40
F    1654     0.05     5.40
F    1655     0.05     5.40
F    1656     0.06     5.20
```

VARIABLE DESCRIPTIONS FOR THE INFIL.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

ABSTR - (r) Global floodplain rainfall abstraction or interception (inches or mm). See comments 1 and 7.

ABSTRINF(N) - (r) Individual floodplain grid element rainfall abstraction (inches or mm).

DTHETA(N) -(r) Individual floodplain grid element soil moisture deficit (SATF-SATI) expressed as a percent. Range (0.0 to 1.0).

HYDC(N) - (r) Average hydraulic conductivity of an individual floodplain grid element (inches/hr or mm/hr). See comments 2, 4 and 5.

HYDCALL - (r) Average global floodplain hydraulic conductivity (inches/hr or mm/hr).

HYDCHN - (r) Average global hydraulic conductivity for the entire channel (inches/hr or mm/hr) to simulate channel seepage. See comment 8.

HYDCONCH(N) - (r) Hydraulic conductivity for a channel element (in/hr or mm/hr).

INFCHAN - (i) Switch 0 or 1; INFCHAN = 1 for channel infiltration (see comment 6).

INFGRIDCH(N) - (i) Array of channel elements with a unique hydraulic conductivity.

INFGRIDFP(N) - (i) Array of floodplain grid elements with individual infiltration parameters (see comment 3).

INFILCHAR(N) – Character identifier 'F' for floodplain (Line 3) or 'C' (Line 5) for channel infiltration data.

POROS - (r) Global floodplain soil porosity. The practical range for soil porosity is $0.30 < POROS < 0.50$.

RTIMPF(N) - (r) Individual floodplain grid element impervious area (ft^2 or m^2).

SATF - (r) Global final saturation of the soil (in percent) for computing infiltration. A practical range for the final saturation is $0.7 < SATF < 1.0$.

SATI - (r) Global initial saturation of the soil (in percent). A practical range is $0.5 < SATI < 1.0$.

SOILS(N) - (r) Capillary suction head for a floodplain grid element (inches or mm).

SOILALL - (r) Average global floodplain capillary suction head (inches or mm).

Instructional Comments for the INFIL.DAT File

1. The infiltration parameters including hydraulic conductivity HYDC, initial abstraction ABSTR, initial saturation SATI, and soil capillary suction head SOILSUC can be estimated from the tables in the User's Manual. 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 infiltration when mudflows are occurring.

3. The individual floodplain grid elements with unique infiltration parameters are specified in Line 3. Line 3 infiltration parameters supercede the values in Line 2.

4. No infiltration is computed for the portion of the grid element removed from the potential flow surface with an Area Reduction Factor (ARF). No infiltration is computed for grid elements that are completely removed from the potential flow surface (ARF = 1.0). Rainfall, however, is assumed to occur over the entire grid element even if the element is completely blocked out by setting ARF = 1. Increased runoff resulting from proposed development can be predicted by using the ARF values to limit infiltration on a grid element.

5. No infiltration is computed for street areas of a grid element. The street area is subtracted from the overland portion of the grid system.

6. Channel infiltration is computed only if INFCHAN = 1. Generally channel infiltration is negligible for channels with perennial flow. The simulation of channel infiltration may be important for small flood events in ephemeral alluvial fan channels with porous bed material.

7. Precipitation abstraction is an initial loss of rainfall that precedes infiltration and excess rainfall runoff. Vegetation interception is a component of the initial loss. Abstraction values will generally range from 0.01 to 0.5 inches. In addition, FLO-2D does not initiate any flood routing until the depression storage TOL is filled. The TOL value is specified in TOLER.DAT file. Abstraction is often assumed to include depression storage, but in FLO-2D a value of 0.1 ft is generally assumed for TOL. This TOL depression storage should be considered when estimating the abstraction.

8. Use HYDCHN on Line 4 to specify uniform channel infiltration. Use Line 5 HYDCON parameter to specify spatially variable hydraulic conductivity in the channel grid elements that will supercede the HYDCHN value in line 4. It is not necessary to specify the soil suction, initial or final saturation values when simulating channel infiltration. These values are assumed not to be important to the channel bed or bank infiltration.

FILE: EVAPOR.DAT
EVAPORATION VARIABLES

**** NOTE: IF IEVAP = 0 IN THE CONT.DAT FILE, OMIT THIS FILE ****

Line 1 has the month, day and time of the start of the simulation to track the hourly evaporation in the model.

1 **IEVAPMONTH, IDAY, CLOCKTIME**

Line 2 is name of month and the monthly evaporation rate.

2 **EMONTH(I), EVAP(I); I=1, 12**

Line 3 is the hourly percentage of the daily evaporation total.

3 **EVAPER(I,J); I=1,12 Months, J=1,24 hours**

**** NOTE: LINE 3 IS REPEATED 24 TIMES FOR EVERY LINE 2 ****

An example of the EVAPOR.DAT is included in the FLO-2D subdirectory base on available data in the Middle Rio Grande valley.

VARIABLE DESCRIPTIONS FOR THE EVAPOR.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

CLOCKTIME - (r) Starting clock time (hrs) of the simulation during the day (range: 0.0 – 24.0).

EMONTH(I) - Name of month for user identification purposes only.

EVAP(I) - (r) Monthly evaporation rate (in/month or mm/month)

EVAPER(I,J) - (r) Hourly percentage of the daily total evaporation (range: 0.00 – 1.00) for each month. There will be 24 values that will total 1.00 for each of the twelve months

IEVAPMONTH - (i) Starting month of simulation (range: 1 – 12).

IDAY - (i) Starting day of the month.

FILE: CHAN.DAT
CHANNEL VARIABLES

**** NOTE: IF ICHANNEL = 0 IN THE CONT.DAT FILE, OMIT THIS ENTIRE FILE ****

Line 1 contains the segment control parameters including the bed elevation switch, initial depth, limiting Froude number, depth roughness adjustment and sediment transport switch.

1 **ICHDEP(K), DEPINITIAL(K), FROUDC (K), ROUGHADJ(K), ISEDN(K), K = 1, number of channel segments**

**** NOTE: LINE 1 IS REPEATED FOR EACH CHANNEL SEGMENT ****
**** NOTE: IF ISED IN CONT.DAT = 0, OMIT ISEDN IN LINE 1****

Line 2 contains all the channel geometry parameters.

**** LINE 2 IS REPEATED FOR EACH CHANNEL GRID ELEMENT ****
**** CHOOSE EITHER 2a, 2b, 2c or 2d in the box below FOR LINE 2 ****

**** IF ICHDEP(K) = 0 OR 1 OMIT BANKELR AND BANKELL IN THE FOLLOWING LINES ****   **** IF ICHDEP(K) = 2 INCLUDE BANKELR AND BANKELL IN THE FOLLOWING LINES ****
Use Line 2a for **rectangular** channels (Set SHAPE = 'R').
2a **SHAPE, ICHANGRID(I), IQDIR(I), BANKELL(I), BANKELR(I), FCN(I), FCW(I), FCD(I), XLEN(I)**
Use Line 2b for **variable area** channels (Set SHAPE = 'V').
2b **SHAPE, ICHANGRID(I), IQDIR(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)**
**** IF EXCDEP(I) = 0, OMIT A11(I), A22(I), B11(I), B22(I), C11(I), C22(I) ****
Use Line 2c for **trapezoidal** channels (Set SHAPE = 'T').
2c **SHAPE, ICHANGRID(I), IQDIR(I), BANKELL(I), BANKELR(I), FCN(I), FCW(I), FCD(I), XLEN(I), ZL(I), ZR(I)**
Use Line 2d for channel cross sections (Set SHAPE = 'N').
2d **SHAPE, ICHANGRID(I), IQDIR(I), FCN(I), XLEN(I), NXSECNUM(I)**

where I = 1, number of channel grid elements in each river segment.

Line 3 is the grid element and water surface elevations for an initial flow in a channel segment.

3a **ISTART, WSELSTART**
3b **IEND, WSELEND**

Line 4 is the channel grid elements that are contiguous but do not share channel discharge.

4 **NOFLO1(N), NOFLO2(N); N = 1, number of NOFLOCs**

FILE: CHAN.DAT (cont.)
CHANNEL VARIABLES

A partial example of the CHAN.DAT file showing the Rio Grande cross section channel geometry data:

```
      2   0.00    0.50    0.20       0
N      60   1 0.031   505.00     1
N      59   1 0.031   510.00     2
N      58   1 0.031   515.00     3
N      57   8 0.031   550.00     4
N      71   2 0.031   620.00     5
N      82   2 0.031   650.00     6
N      93   2 0.031   570.00     7
N     102   2 0.031   660.00     8
N     112   2 0.031   620.00     9
N     124   2 0.031   680.00    10
N     136   2 0.031   630.00    11
            *
            *
            *
N   29955   4 0.030   560.00 1634
N   29968   4 0.030   540.00 1635
N   29977   4 0.030   520.00 1636
N   29984   4 0.029   522.00 1637
      2   0.00    1.20    0.40       1
N    7346   7 0.030   500.00 1638
N    7345   1 0.030   500.00 1639
N    7344   1 0.020   600.00 1640
N    7343   1 0.020   600.00 1641
N    7308   1 0.020   700.00 1642
N    7273   1 0.020   700.00 1643
N    7237   1 0.045   700.00 1644
N    7201   1 0.040   700.00 1645
N    7200   1 0.045   700.00 1646
 26851  26861
 26888  26895
 27643  27670
 25158  25164
 25165  25170
 25170  25176
 25686  25692
 25687  25693
 25692  25699
```

VARIABLE DESCRIPTIONS FOR THE CHAN.DAT FILE

(i) = Integer number (r) = Real number

Variable	Description
A1(I) -	(r) Array of coefficients for the variable area regression relationships.
A2(I) -	(r) Array of exponents for the variable area regression relationships.
A11(I) -	(r) Array of coefficients for the variable area regression relationships for flow depth above EXCDEP(I).
A22(I) -	(r) Array of exponents for the variable area regression relationships for flow depth above EXCDEP(I).
B1(I) -	(r) Array of coefficients for the variable wetted perimeter relationships.
B2(I) -	(r) Array of exponents for the variable wetted perimeter relationships.
B11(I) -	(r) Array of coefficients for the variable wetted perimeter relationships for flow depth above EXCDEP(I).
B22(I) -	(r) Array of exponents for the variable wetted perimeter relationships for flow depth above EXCDEP(I).
BANKELR(I) -	(r) Right bank elevation looking downstream. This variable is required when ICHDEP = 2.
BANKELL(I) -	(r) Left bank elevation looking downstream. This variable is required when ICHDEP = 2.
C1(I) -	(r) Array of coefficients for the variable top width relationships.
C2(I) -	(r) Array of exponents for the variable top width relationships.
C11(I) -	(r) Array of coefficients for the variable top width relationships for flow depth above EXCDEP(I).
C22(I) -	(r) Array of exponents for the variable top width relationships for flow depth above EXCDEP(I).

VARIABLE DESCRIPTIONS FOR THE CHAN.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable **Description**

DEPINITIAL (K) - (i) Initial flow depth for the all channel elements in the given channel segment. If there is no initial channel flow depth set DEPINITIAL = 0. If the initial flow depth is used to specify inflow to the channel, set DEPINITIAL = -1 and include Line 4 (see comment 2).

EXCDEP(I) - (r) Array of channel depths above which a second variable area relationship will apply (see comment 4). If only one channel geometry relationship is used, set EXCDEP(I) = 0.

FCN(I) - (r) Average Manning's n roughness coefficient for the portion of the channel in the grid element ICHANGRID (see comment 6).

FCD(I) - (r) Thalweg channel depth (ft or meters) for the portion of channel within the grid element. The thalweg depth is the deepest part of the channel measured from the lowest top of bank.

FCW(I) - (r) Average channel width (ft or meters). If IAOPT(I) = 0, then FC(I,2) is the rectangular channel width; If IAOPT(I) = 2, then FC(I,2) is the bottom width of a trapezoidal channel. If IAOPT(I) = 1, then the top width is calculated by C1(I)*FCD(I)**C2(I).

FROUDC(K) - (r) Maximum Froude number for main channel flow. If the Froude number exceeds FROUDC, the Manning's n roughness parameter is increased by 0.001. Set FROUDC = 0 for no adjustments of the n-value in a given reach.

ICHDEP(K) - (i) Switch, 0,1 or 2; This variable sets the channel bed elevation (see Figure 1 at the of this section).

ICHDEP = 0, subtract channel depth FCD(I) from grid element floodplain elevation FP(I,6) to determine the channel bed elevation.

ICHDEP = 1, the channel depth is not subtracted from the grid floodplain elevation. The channel bed elevation is equal to node elevation in the FPLAIN.DAT file. The floodplain elevation is then set to the sum of the channel depth and FPLAIN.DAT elevation.

ICHDEP = 2, the channel bed elevation is specified by subtracting channel depth from the bank elevation, either BANKELL or BANKELR (see comment 1). The floodplain is reset to the bank elevation of the channel in the grid element.

VARIABLE DESCRIPTIONS FOR THE CHAN.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable **Description**

ICHANGRID - (i) Grid element containing a channel length (see comments 3, 11 and 14).

IEND - (i) Last channel element for which a starting water surface elevation is specified.

IQDIR - (i) IQDIR is the channel extension direction. If channel width exceeds the grid element, the left bank (looking downstream) will be located in the ICHANGRID node and the right bank will be assigned to a grid element in the IQDIR direction (see comments 5 and 10). Additional elements needed to contain the channel are automatically computed. If the channel topwidth is less than the grid element width, set IQDIR to a direction that the channel could extend. If it is certain that the channel topwidth is less than the grid element width and the channel bank elevations are identical, IQDIR can be set to zero. Specify IQDIR as one of the following directions:

IQDIR = 1 extends the channel to the north,
IQDIR = 2 → east,
IQDIR = 3 → south,
IQDIR = 4 → west,
IQDIR = 5 → northeast,
IQDIR = 6 → southeast,
IQDIR = 7 → southwest,
IQDIR = 8 → northwest

ISEDN(K) - (i) Switch 0 or 1; Set ISEDN(K) = 1 to compute sediment transport in the channel segment and set ISED = 1 in the CONT.DAT file.

ISTART - (i) First channel element for which a starting water surface elevation is specified. DEPINITIAL must be set as a negative number to invoke this routine.

NOFLO1 - (i) Channel element that does not exchange flow with grid element NOFLO2 (see comment 7).

NOFLO2 - (i) Channel element that does not exchange flow with grid element NOFLO1.

NXSECNUM - (i) Cross section number assigned in the XSEC.DAT file. This variable is used only for the cross section data option (see comment 12).

ROUGHADJ – (r) A coefficient used in the depth adjustment of the Manning's n-value (see comment 15). Range: 0.00 – 1.2.

VARIABLE DESCRIPTIONS FOR THE CHAN.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable	Description
SHAPE -	Character line identifier (see comment 4); SHAPE = 'R', rectangular channel geometry (width and depth are input data). SHAPE = 'V', variable area channel geometry (power relationships are used). SHAPE = 'T', a trapezoidal channel (use bottom width, depth and side slopes). SHAPE = 'N', channel cross sections (use cross section survey data)
WSELEND -	(r) Ending water surface elevation for the channel element IEND.
WSELSTART -	(r) Starting water surface elevation for the channel element ISTART.
XLEN(I) -	(r) Array of channel length contained within the grid element ICHANGRID (ft). If more than one channel segment (flow direction) exists in a given grid element, assign XLEN(I) equal to the average representative flow length in one direction (see comments 8, 9 and 13).
ZL(I) -	(r) When IAOPT(I) = 2, ZL(I) is the left side slope of the trapezoidal channel.
ZR(I) -	(r) When IAOPT(I) = 2, 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 (see Figure 1).

2. When DEPINITIAL > 0, an initial depth is specified for all the elements in that channel segment. Setting DEPINITIAL as a negative value will assign starting and ending water surface elevations (WSELSTART AND WSELEND) for the channel segment beginning with element ISTART and ending with element IEND. The water surface elevations are assigned to the rest of the channel elements in that reach 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 with the inflow grid element being the first grid element in the file or segment. 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 continuous in each segment.

4. If channel geometry is being simulated with regression relationships (SHAPE = 'V'), then the area (A) versus depth (d) power relationships ($A = ad^b$) coefficients (a) and exponents (b) must be specified. Similar relationships are required for wetted perimeter and top width. 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. Some rules for applying the second variable area channel geometry relationships follow:

   a. 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.

   b. 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.

   c. 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, B1, B2, C1, C2) in Line 2b.

Instructional Comments for the CHAN.DAT File (cont.)

6. A cross section width can exceed the width of the grid element. For example, a channel cross section 1,000 feet wide can be used in a 200 ft grid system. The number of grid elements required by a cross section is automatically computed. If the channel width exceeds 95% of the grid element width, the model determines the number of grid elements necessary to contain the channel width. If you want to contain the channel within one grid element, the channel top width must be reduced or the grid element size must be increased. The ERROR.CHK file will report missing IQDIRs.

7. Set the channel roughness to a reasonable n-value and then set the FROUDC variable to an appropriate value (e.g. 0.99 to ensure subcritical flow). FLO-2D will adjust the roughness values according to the limiting Froude number criteria (see the ROUGH.OUT file).

8. Channel elements that are contiguous but do not share discharge (e.g. parallel channels) must be identified with the NOFLO1 and NOFLO2 variables. List each pair of contiguous channel elements only once.

9. CADD programs are often imprecise when interpolating between topographic contours to assign node elevations and as a result the channel slope may be erratic. 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 (IAOPT = 3). When one cross section is assigned to several grid elements it will be necessary to interpolate both the slope and the cross section geometry in the PROFILES program to create a smooth average channel slope. Review the PROFILES program instructions for cross section and channel bed slope interpolation. If there is more than one cross section per channel element, use the one that provides the greatest 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. It may be required to do this anyway because the channel is wider than the grid element size.

13. The first two channel elements in a segment should have a positive slope in the downstream direction. This is important for inflow channel elements. There should also be a positive slope into the channel outflow nodes. This will enhance the numerical stability around the inflow and outflow nodes.

Instructional Comments for the CHAN.DAT File (cont.)

14. If you delete a channel element and cross section, you must remove the cross section for that channel element from the XSEC.DAT file. If cross sections are mixed with other channel geometry, the cross section elements can be grouped into segments to identify the reaches with similar channel geometry.

15. Eliminate channel elements that have a XLEN less than 50% of the SIDE (grid element width). This can be accomplished by connecting the channel elements across the diagonal and eliminating the middle channel element.

16. For channel flow, always use the full dynamic wave (KMODEL = 3) in the TOLER.DAT.

17. To improve the timing of the floodwave progress through the system, a depth variable roughness can be assigned on a reach basis. The basic equation for the channel element roughness n_d as function of flow depth is:

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

where:

n_b = bankfull discharge roughness
depth = flow depth
dmax = bankfull flow depth
r2 = roughness adjustment coefficient prescribed by the user (0. to 1.2)
$r_c = 1./e^{-r2}$

This equation prescribes that the variable depth channel roughness is equal to the bankfull roughness at bankfull discharge. If the user assigns a value (from 0 to 1.2) to the roughness adjustment coefficient r2 for a given reach, the roughness will increase with a decrease in flow depth. The higher the coefficient, the greater the increase in roughness. 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 r2 is set to 0.444, the hydraulic roughness Mannings n-value will be 1.4 times the roughness prescribed for bankfull flow.

18. Instructions for creating the cross section channel geometry data files. The FPLAIN.DAT, CADPTS.DAT, CONT.DAT and TOLER.DAT should be completed first.

    a. Create the XSEC.DAT file as prescribed in the next data file format description.

    b. Create the CHAN.DAT file. For the cross section number (NXSECNUM in CHAN.DAT) assign one of the cross section XSECNO to each channel grid element. Some cross sections may be repeated several times. For example, maybe there are 311 channel elements and 30 cross sections. Each channel element NXSECNUM will have a number from 1 to 30. Select the cross section for the grid element on the basis on

topwidth or other similar geomorphic characteristics. Use the GDS, FLOENVIR or hardcopy map or photo with the grid system overlaid to identify the cross sections with respect to the grid element. Consider the need for extending the channel through several floodplain elements. If the cross section has been surveyed from left to right looking downstream, choose a channel grid element to represent the left bank and assign an appropriate channel extension direction IQDIR (CHAN.DAT file) to extend the channel. Avoid having the channel extensions overlap or intersect.

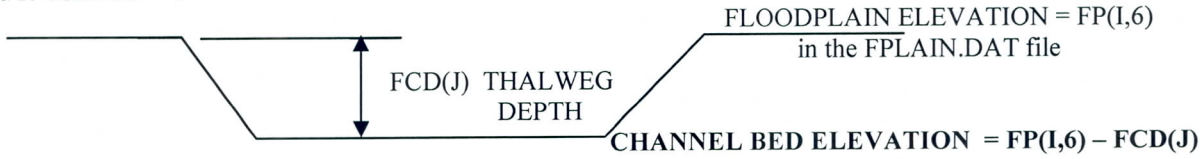
c. Once the CHAN.DAT and XSEC.DAT files are completed, the next step is to create a **unique** cross section for each channel element. Open the PROFILES program, select a reach, zoom-in on a local reach, click on menu command 'Edit Bed Elevations', and then click on the XSEC button in the dialog box. This will display the individual cross section plot and the data for editing. Click 'OK' at the bottom of the dialog box, save the files and exit the program. There will now be the same number of cross sections in the XSEC.DAT data as there as channel grid elements in the CHAN.DAT file (311 channel elements and cross sections). Each channel element now has its own cross section. The original 30 cross sections has been expanded to 311. Obviously, a large number of these cross sections will be identical.

d. Now you can edit and interpolate the channel cross section data. Open PROFILES again and access the cross section plots. The channel profile will look like a stair case until you perform the slope interpolation. Make a list of the channel elements with the original 30 surveyed cross sections. You can mark these in the XSEC.DAT file by placing an X after the cross section name (e.g. CO-413 becomes CO-413X). If you edit the cross section names in PROFILES, click 'OK' in the dialog box to accept the name change. This task can also be performed in MS WordPad. You can now edit cross section data, delete data, raise or lower the bed or add points to the cross section. The most important task you will perform is the interpolation of the cross sections between known surveyed cross sections. Either the slope or the combined slope and cross section geometry can be interpolated. Simply enter the number of the two channel grid elements with the upstream and downstream surveyed cross sections (cross section names with the X) in the dialog box and click the appropriate interpolation button. New cross sections will be interpolated for all the channel elements between the two surveyed cross sections (exclusive). This is a flow area weighted interpolation of the cross section shape. The results will be an individually modified cross section for each channel grid element and the channel profile will have a gradual slope not a staircase. Save the files and exit the PROFILES program.

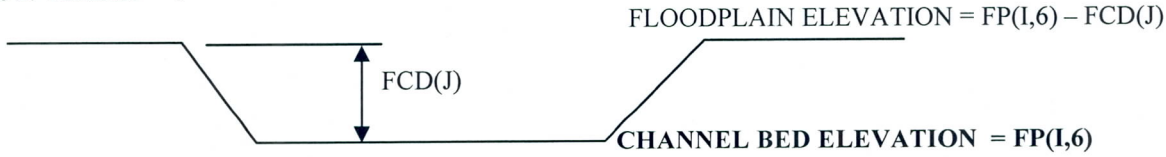
It is recommended that you save the files and make backups frequently until you are familiar with the procedure.

CHANNEL CROSS SECTION CONFIGURATIONS

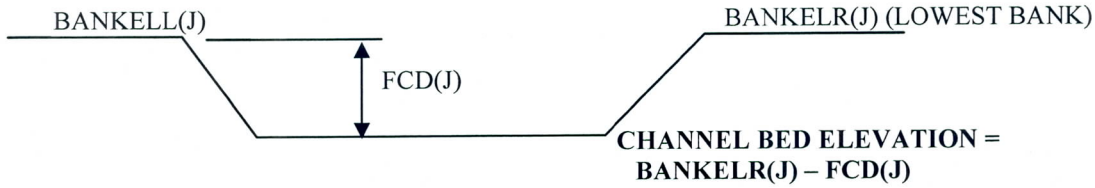
Case 1. ICHDEP = 0



Case 2. ICHDEP = 1

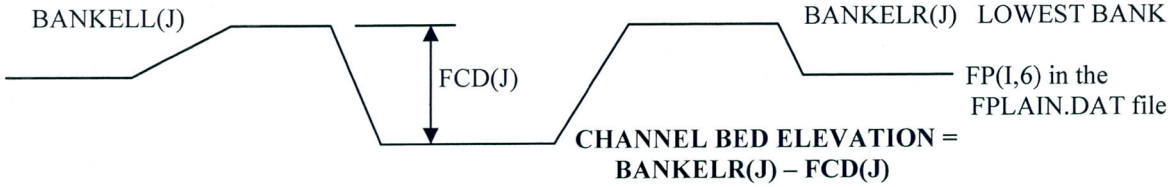


Case 3. ICHDEP = 2

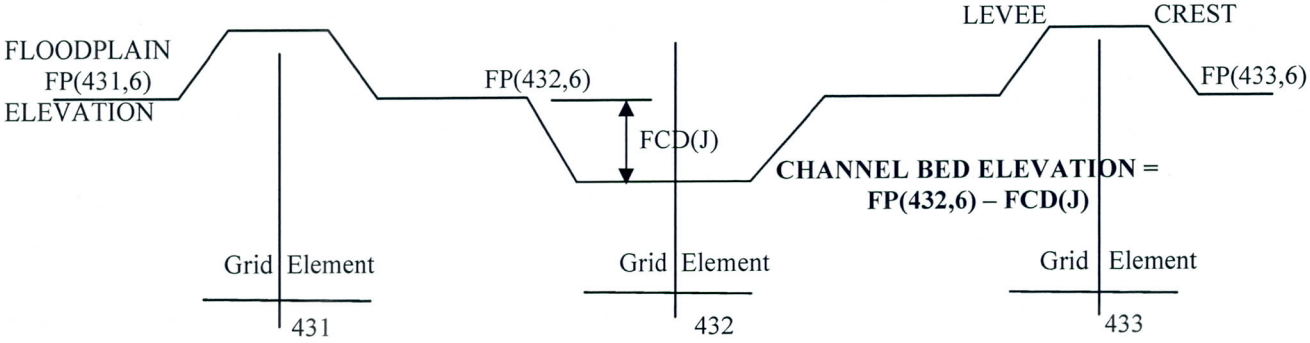


EXAMPLE APPLICATIONS

USE ICHDEP = 2 FOR THIS CONFIGURATION



USE A LEVEE AND ICHDEP = 0 FOR THIS CONFIGURATION



FILE: XSEC.DAT
CHANNEL CROSS SECTION DATA

**** NOTE: IF ICHANNEL = 0 IN THE CONT.DAT FILE, OMIT THIS FILE ****
**** NOTE: SET SHAPE = 'N', LINE 2d IN CHAN.DAT TO USE THIS FILE ****

Line 1 Cross section character identifier, cross section number and cross section ID name.

1 **XSECHAR = 'X', XSECNUM(I), XSECNAME(I); I = 1, number of cross sections**

**** NOTE: LINE 1 IS REPEATED FOR EACH CROSS SECTION ****

Line 2 x-station, y-elevation from left bank looking downstream.

2 **XI(I,J), YI(I,J); I = 1, number of cross sections, J = 1, number of pairs of station and elevation points**

**** NOTE: LINE 2 IS REPEATED FOR THE NUMBER OF STATION AND ELEVATION PAIRS IN EACH CROSS****

A partial example of a XSEC program data file follows:

```
X      1  CI-27.1
     0.0  5235.07
    10.0  5235.17
    25.0  5235.31
    30.0  5231.84
    35.0  5230.51
    40.0  5229.73
    50.0  5229.08
    60.0  5228.88
    70.0  5228.58
    80.0  5228.58
    90.0  5228.38
   100.0  5227.98
   110.0  5227.88
          *
          *
          *
   274.0  5228.58
   283.0  5229.38
   281.0  5229.84
   285.0  5233.07
   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  5229.42
    50.0  5228.77
    60.0  5228.57
```

VARIABLE DESCRIPTIONS FOR THE XSEC.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

XI(I,J) - (r) Array of cross section station distances from the left end point (ft or m). The initial value of XI can start at any value greater than 0.0.

XSECHAR – (a) Character 'X' that identifies Line 1.

XSECNAME(I) - (a) Cross section name (less than 10 characters). This name is for cross section ID purposes only, it is not used by the model.

XSECNUM(I) – (i) Cross section number starting with 1 and ending with the last surveyed cross section.

YI(I,J) - (r) Array of cross section elevations (ft or m) at each station. The value of YI can be an actual elevation or any value greater than 0.0.

FILE: HYSTRUC.DAT
HYDRAULIC STRUCTURE VARIABLES

**** NOTE: IF IHYDRSTURCT = 0 IN THE CONT.DAT FILE, OMIT THIS FILE ****

Line 1 lists the character indicator for Line 1, structure name, choice of floodplain or channel element, inflow and outflow nodes, choice of inlet or outlet control, and reference elevation for structure.

1 **STRUCHAR = 'S', STRUCTNAME, IFPORCHAN(I), ICURVTABLE(I), INFLONOD(I), OUTFLONOD(I), INOUTCONT(I), HEADREFEL(I)**

**** NOTE: IF A RATING CURVE IS USED (ICURVTABLE(I) = 0), INCLUDE LINE 2 ****

Line 2 is the discharge-headwater depth relationships for the structure rating curves including character indicator for Line 2, headwater depth, rating curve coefficient and exponent and a replacement rating curve switch.

2 **STRUCHAR = 'C', HDEPEXC(I,J), COEFQ(I,J), EXPQ(I,J); I = 1, number of structures; J = 1, number of rating curves**

****NOTE: LINE 2 IS REPEATED FOR EACH RATING CURVE ****

**** NOTE: INCLUDE LINE 3 IF A REPLACEMENT RATING CURVE IS REQUIRED ****

Line 3 is the structure replacement rating curve to simulate flow blockage or a change in the rating curve during the flood simulation. It includes character indicator for Line 3, headwater depth to initiate the change in the rating curve, the new rating curve coefficient and exponent.

3 **STRUCHAR = 'R', REPDEP(I,J), RQCOEF(I,J), RQEXP(I,J); I = 1, number of structures, J = 1, IREPLACE(I,J)**

****NOTE: IF A RATING TABLE IS USED (ICURVTABLE(I) = 1), INCLUDE LINE 4 ****

Line 4 is the rating table character indicator, depth and discharge for a given structure.

4 **STRUCHAR = 'T', HDEPTH(I,J), QTABLE(I,J); I = 1, number of structures; J = 1, number of pairs of data.**

****NOTE: LINE 4 IS REPEATED NDPAIRS TIMES ****

****NOTE: ONLY ONE SET OF LINES 1, 2 AND 3* ***OR*** *ONE SET OF LINES 1 AND 4 ****
**** SHOULD BE INCLUDED FOR EACH HYDRAULIC STRUCTURE ****

FILE: HYSTRUC.DAT (cont.)
HYDRAULIC STRUCTURE VARIABLES

An example of the HYSTRUC.DAT file with three structures follows:

```
S Patagonia  1  0  1713 1827 0   4425.23
C    20.   3.543  0.890
S Mayberry  1  1  1856  1945  0   4421.18
T      0.       0.
T      5.       250.
T      8.5      500.
T      10.      1000.
T      12.5     1500.
T      15.0     2500.
T      17.0     3000.
S Keitzer  0  0   2503   2725  1  0.0
C   5.    25.023    1.035
C  10.  30.00       1.4
R  12.   0.00       1.0
```

VARIABLE DESCRIPTIONS FOR THE HYSTRUC.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

COEFQ(I,J) - (r) Array of hydraulic structure discharge rating curve coefficient where the discharge is expressed as a power function of the headwater depth. Q = COEFQ(I,J) * depth$^{EXPQ(I,J)}$ (see comment 1). If COEFQ(I,J) = 0, then the discharge is computed as normal depth flow routing.

EXPQ(I,J) - (r) Array of hydraulic structure discharge exponents where the discharge is expressed as a power function of the headwater depth.

HDEPEXC (I,J) – (r) Maximum depth that a hydraulic structure rating curve is valid (ft or m).

HDEPTH(I,J) - (r) Array of headwater depths for the structure headwater depth-discharge rating table (ft or m). See comments 2 and 4.

HEADREFEL(I) - (r) 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.

ICURVTABLE(I) - (i) Switch, either 0 or 1. Set ICURVTABLE(I) = 0 for a structure rating curve. Set ICURVTABLE(I) = 1 for a structure rating table. See comment 6.

IFPORCHAN(I) - (i) Switch, either 0 or 1. Set IFPORCHAN(I) = 0 for a floodplain structure. Set IFPORCHAN(I) = 1 for a channel hydraulic structure.

INFLONOD(I) - (i) Grid element containing the hydraulic structure or structure inlet.

INOUTCONT(I,J) - (i) Switch 0 or 1; Set INOUTCONT(I,J) = 0 for inlet control based on the headwater depth and set INOUTCONT(I,J) = 1 to evaluate tailwater effects. If INOUTCONT(I,J) = 0, the structure discharge is computed based on the headwater depth above the reference elevation. If INOUTCONT(I,J) = 1, the discharge is based on the difference between the headwater and tailwater depth.

OUTFLONOD(I) - (i) Grid element receiving the hydraulic structure discharge (structure outlet). OUTFLONOD does not have to be contiguous to INFLONOD grid element.

QTABLE(I,J) - (r) Array of hydraulic structure discharges for the headwater depths in the rating table (discharge). See comments 3 and 5.

REPDEP(I,J) - (r) 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.

VARIABLE DESCRIPTIONS FOR THE HYSTRUC.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable	Description

RQCOEF(I,J) - (r) Structure rating curve replacement coefficients. There should be the same number of rating curve pairs of coefficients and exponents.

RQEXP(I,J) - (r) Structure rating curve replacement exponents. There should be the same number of rating curve pairs of coefficients and exponents

STRUCHAR - (c) Character that identifies the use of Line 2, 3 or 4 where STRUCHAR = 'S' for the structure control, STRUCHAR = 'C' for a rating curve (Line 2); STRUCHAR = 'R' for replacement rating curve (Line 3); or STRUCHAR = 'T' for a rating table (Line 4).

STRUCTNAME(I) – (i) Hydraulic structure name (15 characters or less). This name is for user identification purposes only.

Instructional Comments for the HYSTRUC.DAT File

1. Hydraulic structure discharge between either floodplain or channel grid elements can be simulated as a rating curve function of the headwater depth, $Q = COEFQ*depth^{EXPQ}$, where COEFQ and EXPQ are specified coefficients and exponents which are valid for a depth not to exceed HDEPEXC. The grid elements containing the structure inlet and outlets must be specified. The inlet and outlet grid elements do not have to be contiguous. The structure discharge (such as a culvert, weir or bridge) may either inlet or outlet control as long as the discharge is specified as power function of the headwater depth.

2. When the headwater depth exceeds the specified depth (HDEPEXC) for which the original rating curve relationship is valid, a second replacement relationship is invoked. These multiple relationships can be used to specify structure blockage or a change in the rating curve. For example, if a height of 5 ft corresponds to the top of a culvert for a discharge of up to 300 cfs, then a second rating curve relationship for flows over a roadway could based on a flow depths starting at 6 ft above the culvert invert that could correspond to a discharge of greater than 500 cfs. Structure blockage can be simulated by setting the replacement coefficient (RQCOEF) equal to zero.

3. If a hydraulic structure rating table is used, a linear interpolation between two headwater depths in the table is applied to estimate the discharge for a headwater depth determined the model.

4. When structure tailwater effects must be considered, the HDEPEXC variable for the rating curves represents the difference between the headwater and tailwater water surfaces.

5. The rating table should always have the first pair of depth-discharge data as headwater depth = 0 and discharge = 0 to enable interpolation with the next data pair in the rating table.

6. The hydraulic structure may be any type of flow control such as a bridge, diversion, culvert, weir, roadway or spillway. If a culvert is simulated that is separated by more than one grid element, neither the travel time or volume of storage in the culvert is considered. The discharge is computed at the outflow element for the same timestep. This is a relatively minor assumption that should not affect your simulation unless the culvert can contain a significant portion of the flood volume in the entire model.

FILE: STREET.DAT

Street Flow Variables

**** NOTE: IF MSTREET IN CONT.DAT IS EQUAL TO ZERO, OMIT THIS FILE ****

Line 1 specifies the street flow control parameters and global assignment of the street variables.

1 **STRMAN, ISTRFLO, STRFNO, DEPX, WIDST**

**** NOTE: LINES 2 THROUGH 4 ARE REPEATED FOR EACH STREET ****

Line 2 contains the name of the street. This street name is not used by the model.

2 **STCHAR = 'N', STNAME**

Line 3 comprises of a list of street variables including a line character identifier, grid element number, curb height, street roughness and street elevation.

3 **STRCHAR = 'S', IGRIDN(L), DEPEX(L), STMAN(L), ELSTR(L); L=1, to the number of grid elements in each street.**

**** NOTE: DEPEX(L), STMAN(L), ELSTR(L) CAN BE SET EQUAL TO 0.0 ****
**** IF THE GLOBAL VALUES IN LINE 1ARE UTILIZED ****

**** NOTE: EACH GRID ELEMENT SHOULD BE LISTED ONLY ONCE IN THE FILE ****

Line 4 lists a line character identifier, street segment direction and width in each grid element.

4 **STRCHAR = 'W', ISTDIR(K), WIDR(K); K=1, to the number of street segments in each grid element (up to 8).**

****NOTE: WIDR(K) CAN SET EQUAL TO 0.0 ****
**** IF THE GLOBAL VALUES IN LINE 1 ARE UTILIZED****

**** NOTE: REPEAT LINE 4 FOR THE NUMBER OF GRID ELEMENT ****
**** STREET SEGMENTS BEFORE REPEATING LINE 3****

A partial example of the STREET.DAT file follows:

```
    0.025        1      1.7    0.667       40
N   MAIN
S              127        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
W                4       50
S              132        0        0        0
W                1       40
W                2       50
W                4       50
S              133        0        0        0
W                2       50
W                4       50
S              134        0        0        0
W                2       50
W                4       50
S              135        0        0        0
W                2       50
W                4       50
S              136        0        0        0
W                2       50
W                4       50
N     ONE
S              183        0        0        0
W                2       90
S              184        0        0        0
W                2       40
W                4       40
S              185        0        0        0
W                2       40
W                4        0
N     TWO
S              224        0        0        0
W                2        0
W                4        0
S              225        0        0        0
W                2        0
W                4        0
S              226        0        0        0
```

VARIABLE DESCRIPTIONS FOR THE STREET.DAT FILE

(i) = Integer variable (r) = Real variable

Variable	Description
DEPX -	(r) Street curb height (ft or m). If the street curb height is exceeded by the flow it will result in overland flow in the grid element containing the street. DEPX is used to assign a street curb height to all grid elements.
DEPEX(L) -	(r) 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) Optional street elevation (ft or m). This elevation will supercede the floodplain grid element elevation. If ELSTR(L) = 0., the model will assign the street elevation as grid element elevation, FP(I,6) minus the curb height DEPEX(L) or DEPX to the street elevation ELSTR(L) (see comment 3).
IGRIDN(L) -	(i) Grid element number. Each grid element should be listed only once in the data file (see comment 6).
ISTDIR(K) -	(i) Street segment (flow direction) from the center of the grid element to a neighboring element. ISTDIR(K) will vary from 1 to 8 according to the following compass directions:

1 = north	5 = northeast
2 = east	6 = southeast
3 = south	7 = southwest
4 = west	8 = northwest.

ISTRFLO -	(i) Switch 0 or 1; where ISTRFLO = 1 specifies that the system inflow hydrograph will enter the streets rather than entering the overland portion of the grid element.
STRCHAR -	Character 'N', 'S' or 'W' to identify either Line 2, 3 or 4.
STRFNO -	(r) 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.
STMAN(L) -	(r) Optional street n-value for the individual street segments within a given grid element. STMAN(L) supercedes the STRMAN value. If STMAN(L) = 0.0, the global value STRMAN will be assigned to the grid element street segment.
STRMAN -	(r) Global n-value for street flow which that is assigned to all the grid element street segments. See comment 2.

VARIABLE DESCRIPTION FOR THE STREET.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable	Description
STNAME -	Character name of the street. Up to 15 characters can be used. The street name is not used in the model (see comment 1).
WIDR(K) -	(r) 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) Global assignment of street width to all streets. This value is superseded by WIDR(K) when WIDR(K) is greater than zero (see comments 2 and 4).

Instructional Comments for the STREET.DAT FILE

1. The street name is provided for the user to separate the streets groups for easy identification in the data file. It is not used in the program.

2. The street depth, width and n-values can be assigned globally for all the street grid elements. The street depth, width and elevation can be uniquely assigned for individual grid elements in Lines 3 and 4 by setting these values greater than zero.

3. If the street elevation is different from the representative grid elevation assigned in the FPLAIN.DAT file, it should be specified in line 3, otherwise the street elevation will be the floodplain elevation minus the curb height. This elevation is then used to determine the street slope.

4. The street width should be less than the side of the grid element. The overall floodplain surface area of the grid after the streets are removed must be at least 5% of the original surface area (grid element width squared). If there are numerous streets in the grid element that consume all the grid element surface area, consider leaving out the smaller, less significant streets, reduce the street width or transfer one or more to a neighboring grid element. Another option is to increase the grid element size and redo the grid system.

5. The street is assumed to extend from the center of the grid element to the grid element boundary in the four compass directions plus the four diagonal directions as specified by the variable ISTDIR(K). A street that crosses the entire grid element is assigned two street sections and directions in Line 4.

6. Each grid element should be listed only once in the STREET.DAT file. For street intersections within the grid element, list all the street flow directions for the first street, then skip that grid element for the succeeding crossing streets.

FILE: ARF.DAT
AREA AND WIDTH REDUCTION VARIABLES

**** NOTE: IF IWRFS IN THE CONT.DAT FILE EQUALS 0, OMIT THIS FILE ****

Line 1 contains the character identifier and the list of grid elements that are totally blocked out from receiving any flow.

1 **ITTCHAR = 'T', ITTAWF(K), K=1, number of total blocked grid elements**

**** NOTE: LINE 1 IS REPEATED FOR EACH TOTALLY BLOCKED GRID ELEMENT ****

Line 2 contains the grid element and the associated optional area and width reduction factors.

2 **IGD(I), ARF(I), WRF(I,J); I=1, number of partially blocked grid elements; J = 1, 4)**

An partial example of the ARF.DAT file follows:

```
T 49
T 65
T 86
T 87
T 88
T 89
T 102
T 103
T 104
T 105
T 162
   29   .20   .70   .50   1.0    0.
   30   .50   1.0   .80   1.0    0.
   32    0.   .50   .55   1.0    0.
   33   .65   1.0   .35   1.0    0.
   46   .10   .30    0.    0.    0.
   47   .25   .50    0.    0.    0.
   48   .15   .30    1.   1.0   1.0
   61   .05    0.   .30    0.   1.0
   62   .40   .40    0.   0.8   0.0
   63   .15    0.   .07   0.0   0.0
   64   .30   .25   1.0   1.0   1.0
   70    0.    1.    0.   1.0   0.0
   71    0.    1.    0.   1.0   0.0
   72    0.    1.    0.   1.0   0.0
   73    0.    1.    0.   1.0   0.0
   78   .50   .60    0.   0.0   0.0
   81   .40    0.    .9   0.5   0.5
   82   .20   .25    1.   1.0   0.0
   94   .55   .60    .8   1.0   0.0
   95   .15   .90   .85   1.0   0.0
   96   .20   .25   .10   0.8   0.0
   97   .20   .10   .10   0.0   0.5
   98   .25   .05   .50   0.0   0.5
   99   .25   .10   .05   0.0   0.0
  107   .20    0.    .5   0.0   0.0
```

VARIABLE DESCRIPTIONS FOR THE ARF.DAT FILE

(i) = Integer number (r) = Real number

Variable	Description
ARF(I) -	(r) Optional area reduction factor (ARF) for grid element I. The range for ARF(I) is $0. \leq ARF(I) \leq 1$. ARF(I) is the percent of the grid element area that can not be covered by surface flows. Buildings or other physical features contained within the grid that cannot store flow volume are accounted by using the ARF value. The maximum value for an ARF = 0.95. If ARF > 0.95, the ARF is reset to 0.95 (see comments 1 and 3).
IGD(I) -	(i) Array of partially blocked grid element numbers.
ITTAWF(I) -	(i) Array of grid elements that will not receive any flow. Each grid element is totally blocked out and all ARF and WRF values are set equal to 1.0.
ITTCHAR -	Character to identify Line 1. Set ITTCHAR = 'T' to identify Line 1.
WRF(I,J) -	(r) Array of optional width reduction factors (WRF) for grid element(I). The range for WRF(I,J) is $0 \leq WRF(I,J) \leq 1.0$. The width reduction factor corresponds to the percentage of flow width lost due to obstruction in four directions, north, east, northeast and southeast. The width reduction for the other four directions is defined by the values assigned to the contiguous elements. Assuming that the flow field is oriented with the north direction toward the top of the page, use the following WRF assignment: WRF(I,1) = North WRF(I,2) = East WRF(I,3) = Northeast WRF(I,4) = Southeast where I is the grid element number (see comment 2).

Instructional Comments for the ARF.DAT File

1. Ffor a partially blocked grid element, those ARF and WRF values that are 0.0 must be entered. The graphical assignment and editing of ARF and WRF values are easy in either the FLOENVIR or GDS 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. Only four of the eight sides need to be input. The other four sides are determined by the model from the other contiguous grid elements.

3. The maximum ARF value is 0.95 unless the grid element is totally blocked out in Line 2 as a ITTAWF grid element. This insures that at least 5% of the grid element is left for flow storage. For ARF values greater than 0.95, the model will reset the ARF to 0.95. Use the ITTAWF parameter to totally block out a grid element with zero storage.

FILE: MULT.DAT
OVERLAND MULTIPLE CHANNEL VARIABLES

**** NOTE: IF IMULTC IN THE CONT.DAT FILE IS EQUAL TO 0, OMIT THIS FILE ****

Line 1 is the global assignment of multiple channel (rills and gullies) parameters including the incremental width increase, multiple channel width, depth, number of channels per grid element, and n-values.

1 **WMC, WDRALL, DMALL, NODCHNSALL, XNMULTALL**

**** NOTE: IF WDRALL = 0, NO GLOBAL ASSIGNMENT OF THE VARIABLES OCCURS ****

Line 2 contains individual grid element multiple channel parameter revisions.

2 **IGRID(I), WDR(I), DM(I), NODCHNS(I), XNMULT(I); I = 1, end of grid elements with multiple channels**

**** NOTE: LINE 3 IS REPEATED FOR EACH GRID ELEMENT REVISION ****

A partial example of the MULT.DAT files follows:

```
  1      5      1      1   0.060
  2     50      6      1   0.055
 27     20      3      1    0.06
 28     20      3      1    0.06
 29     20      3      1    0.06
 30     20      3      1    0.06
 50     10      2      1    0.06
 51     10      2      1    0.06
 52     10      2      1    0.06
 53     10      2      1    0.06
 54     10      2      1    0.06
 55     10      2      1    0.06
 56     10      2      1    0.06
 57     10      2      1    0.06
 58     10      2      1    0.06
 79      5      1      1    0.06
 80      5      1      1    0.06
 81      5      1      1    0.06
 82      5      1      1    0.06
 83      5      1      1    0.06
 84      5      1      1    0.06
 85      5      1      1    0.06
 86      5      1      1    0.06
 87      5      1      1    0.06
 88      5      1      1    0.06
 89      5      1      1    0.06
 90      5      1      1    0.06
 91      5      1      1    0.06
 92      5      1      1    0.06
 93      5      1      1    0.06
108      3      1      1    0.06
```

VARIABLE DESCRIPTIONS FOR THE MULT.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

DM(K) - (r) Maximum depth of multiple channels for individual grid elements. When the flow depth exceeds the multiple channel depth DM, the flow width WDR of the gully is increased by the incremental width WMC (see comments 2 and 3). DM supercedes the DMALL depth assignment.

DMALL - (r) Global assignment of the maximum depth to all grid elements.

IGRID(I) - (i) Floodplain grid element number (see comment 1).

NODCHNS(K) - (i) Number of multiple channels assigned in each grid element. If NODCHNS is set equal to zero then the overland flow without multiple channels is assumed. NODCHNS supercedes NODCHNSALL value.

NODCHNSALL - (i) Global assignment of the number of multiple channels to all grid elements.

WDR(K) - (r) Channel width for individual grid elements. WDR supercedes WDRALL.

WDRALL - (r) Global assignment of the multiple channel width to all grid elements. If WDRALL = 0, all global variables are set to zero.

WMC - (r) Incremental width by which multiple channels will be expanded when the maximum rill and gully depth DM is exceeded (see comments 2 and 4).

XNMULT(K) - (r) Channel n-values for individual grid elements. Supercedes XNMULTALL.

XNMULTALL - (r) Global assignment of the multiple channel n-values to all the grid elements.

Instructional Comments for the MULT.DAT File

1. If a grid element is assigned multiple channels and, in addition, contains a main channel or buildings, such that the available floodplain surface storage area is less than 50% of the original grid element surface area, then the model will reset that grid element to overland sheet flow (i.e. no multiple channels). Multiple channels will not be simulated in the grid elements with streets. The program will automatically eliminate any multiple channels in grid elements with streets. The user does not have to alter the data files. The available surface area and the assigned variable can be review in the SURFAREA.OUT output file.

2. If a gully channel fills and is about to overflow, it is assumed that the gully is an alluvial channel and will widen to accept more flow. Thus when the flow depth exceeds the maximum gully channel depth DM, the model increases the gully width by the incremental width value WMC. Once the flow is in a gully, the gully will not overflow back onto the overland area, but will continue to widen until the gully is wider than the grid element or until there is no more overland storage area left in the grid element. The flow routing will then revert to overland flow in that element.

3. If it is desired to force the flow to stay in a channel of fixed width, set the variable WMC = 0.

4. The total flow width is determined by multiplying the number of channels in each grid element by the appropriate width.

FILE: SED.DAT
MUDFLOW AND SEDIMENT TRANSPORT VARIABLES

**** NOTE: IF MUD = 0 IN THE CONT.DAT, OMIT LINE 1. ****

Line 1 is the hyperconcentrated sediment flow parameters

1 **VA, VB, YSA, YSB, SGSM, XKX**

**** NOTE: IF ISED IS EQUAL TO 0, IGNORE LINES 2,4,5 and 6 ****

Line 2 lists parameters for sediment routing.

2 **ISEDEQ, DFIFTY, SGRAD, SGST, DRYSPWT, ISEDSUPPLY, ISEDISPLAY**

**** IF IDEBRV IS EQUAL TO 0 IN THE CONT.DAT FILE, IGNORE LINE 3 ****

Line 3 contains debris basin parameters.

3 **JDEBNOD, DEBRISV**

Line 4 represents the scour depth limitation for both channel and floodplain grid elements.

4 **SCOURDEP**

Line 5 contains the list of rigid bed nodes.

5 **SEDCHAR = 'R', ICRETN(N), N = 1, number of rigid bed nodes**

**** IF ISEDSUPPLY IS EQUAL TO 0, IGNORE LINES 6 ****

Line 6 is the grid element, floodplain or channel switch, coefficient and exponent for a sediment supply rating curve.

6 **SEDCHAR = 'S', ISEDGRID(N), ISEDCFP(N), ASED(N), BSED(N), N = 1, number of sediment supply rating curves**

An example of the SED.DAT file for the Rio Grande follows:

```
2   0.25   2.5   2.65    92.5   1   7232
0.
R 9366
R 7346
R 7345
R 7344
R 7343
R 7308
R 7273
R 7237
S 5069  1  0.0047  1.794
S 7165  1  1.13    0.885
```

VARIABLE DESCRIPTIONS FOR THE SED.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

ASED(N) - (r) Sediment rating curve coefficient (see the BSED variable below).

BSED(N) - (r) Sediment rating curve exponent. $Q_s = ASED^* Q_w^{BSED}$, where Q_w is the water discharge (cfs or cms) and Q_s is the sediment supply (tons/day or kg/day).

DEBRISV - (r) Volume of the debris basin in ft^3 or m^3.

DFIFTY - (r) Sediment size (D_{50}) in mm for sediment routing.

DRYSPWT - (r) Dry specific weight of the sediment (lb/ft^3 or N/m^3).

ICRETN (N) - (i) Individual floodplain grid elements with a rigid bed (e.g. spillway apron).

ISEDCFP(N) – (i) Switch, 0 or 1; set ISEDCFP(N) = 0 for a floodplain sediment supply rating curve and ISEDCFP(N) = 1 for a channel sediment supply rating curve.

ISEDEQ - (i) Number that identifies the sediment transport capacity equation to be used in sediment routing. Set ISEDEQ as follows for the appropriate sediment transport equation:

ISEDEQ = 1, Zeller and Fullerton
ISEDEQ = 2, Yang
ISEDEQ = 3, Englund and Hansen
ISEDEQ = 4, Ackers and White
ISEDEQ = 5, Laursen
ISEDEQ = 6, Toffaleti

ISEDISPLAY - (i) Grid element (channel or floodplain) for which the sediment transport capacity for all the sediment transport equations will be displayed by output interval TOUT in the SEDTRAN.OUT file. Please note that only one equation is used in the actual sediment routing calculations, but the results of all equations are displayed in SEDTRAN.OUT.

ISEDGRID(N) – (i) Grid element that will be a sediment supply node (channel or floodplain) with a sediment rating curve.

ISEDSUPPLY - (i) Switch 0 or 1; set ISEDSUPPLY = 1 if a sediment rating curve will be used to define the sediment supply to a channel reach or floodplain area.

JDEBNOD - (i) Grid element with the debris basin.

VARIABLE DESCRIPTIONS FOR THE SED.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable	Description
NSUPPLY -	(i) Number of sediment rating curves to define the sediment supply to the model.
SEDCHAR -	Character identifier for Lines 5 and 6. Set SEDCHAR = 'R' to input rigid bed grid elements. Set SEDCHAR = 'S' to input sediment supply rating curves.
SCOURDEP	(i) Maximum allowable scour depth (ft. or meters) for all floodplain elements.
SGRAD -	(r) Sediment gradation coefficient (non-dimensional) for the sediment transport routine.
SGSM -	(r) Specific gravity of the sediment in the mudflow.
SGST -	(r) Specific gravity of the sediment for sediment transport.
VA -	(r) 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.
VB -	(r) Exponent in the viscosity versus sediment concentration by volume relationship.
XKX -	(r) The laminar flow resistance parameter for overland flow. This value should range from 24 to 50,000 (see page 36 of the front part of the manual). It is suggested that a value of 2480 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 Mannings n-value:

$$\begin{aligned} FPN < 0.01, &\quad XKX = 24. \\ 0.01 < FPN < 0.25, &\quad XKX = 1{,}460{,}865.81 * (FPN)^{2.381} \\ 0.25 < FPN, &\quad XKX = 2480 \end{aligned}$$

YSA -	(r) Coefficient of the yield stress versus sediment concentration by volume relationship. The relationship is based on a yield stress given in dynes/cm^2 for either the English or Metric system.
YSB -	(r) Exponent of yield stress versus sediment concentration by volume relationship.

FILE: LEVEE.DAT
LEVEE AND LEVEE FAILURE VARIABLES

**** NOTE: IF LEVEE = 0 IN THE CONT.DAT FILE, OMIT THIS FILE ****

Line 1 is the incremental height to increase levee crest elevation.

1 **RAISELEV**

Line 2 lists a line character identifier and levee grid element number.

2 **LEVCHAR = 'L', LGRIDNO(L); where L = 1, number of levee grid elements**

Line 3 contains a line character identifier, the levee blockage direction (8 possible directions) and the crest elevation for each direction.

3 **LEVCHAR = 'D', LDIR(L,J), LEVCREST(L,J); where J = 1, number of levee directions in the grid element**

Line 4 has a line character identifier and the levee failure grid element.

4 **LEVCHAR = 'F', LFAILGRID(LF); where LF = 1, number of levee failure grid elements**

**** NOTE: LINE 4 IS REPEATED FOR EACH LEVEE FAILURE GRID ELEMENT ****

Line 5 lists the levee failure data including a line character identifier, the prescribed levee failure direction, potential failure elevation, failure duration time, fail-to-base level elevation, rate of increase in failure breach width and rate of increase in vertical beach failure.

5 **LEVCHAR = 'W', LFAILDIR(LF,LD), FAILEVEL(LF,LD), FAILTIME(LF,LD), LEVBASE(LF,LD), FAILWIDTH(LF,LD), FAILRATE(LF,LD), FAILWIDRATE(LF,LD); where LD = 1, number of failure directions and LF = 1, number of levee failure grid elements**

**** NOTE: LINE 5 IS REPEATED FOR EACH GRID ELEMENT FAILURE DIRECTION****

An example of a partial LEVEE.DAT file follows:

```
    0.00
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    1952
D       1    5020.00
D       4    5020.00
D       8    5020.00
L    1953
D       8    5020.00
L    1959
D       2    5020.00
D       5    5020.00
D       6    5020.00
```

VARIABLE DESCRIPTIONS FOR THE LEVEE.DAT FILE

(i) = Integer number (r) = Real number

Variable **Description**

FAILEVEL(LF,LD) - (r) The maximum elevation of the prescribed levee failure if different (lower) than the levee crest (LEVELEV). Set FAILEVEL = 0 to fail the levee when overtopped.

FAILRATE(LF,LD) - (r) The rate of vertical levee failure in feet or meters per hour.

FAILTIME(LF,LD) - (r) The duration (hours) that the levee will fail after the FAILEVEL elevation is exceeded by the flow depth. Set FAILTIME = 0 if the level fails immediately when overtopped.

FAILWIDRATE(LF,LD) - (r) The rate at which the levee breach widens (ft/hr or m/hr). The breach stops increasing if the breach exceeds the grid element width for that direction.

FAILWIDTH(LF,LD) - (r) The flow width (ft or m) of levee failure. This flow width relates to one of the eight flow directions and should be less than the length of the side of a grid element times 0.4142 (length of an octagon side).

LDIR(L,IM) - (i) Flow direction (of the 8 possible overland flow directions) that will be cutoff by a levee in a given grid element. The possible flow directions are:

1 = north	5 = northeast
2 = east	6 = southeast
3 = south	7 = southwest
4 = west	8 = northwest

LEVBASE(LF,LD) - (r) The elevation of the levee crest after complete failure. Set LEVBASE = 0 if levee failure results in the levee crest being reset to the floodplain elevation in the FPLAIN.DAT file. If levee failure exceeds LEVBASE elevation, the floodplain elevation is set to LEVBASE.

LEVCHAR(L) - Character Identifier for Lines 2 and 3, where 'L' = line 2, 'D' = line 3, 'F' = line 4, and 'W' = line 5.

LEVCREST(L,IM) - (r) The elevation of the levee crest (see comment 4).

LFAILDIR(LF,LD) - (i) The potential failure direction (one of eight possible flow directions). See comment 3.

VARIABLE DESCRIPTIONS FOR THE LEVEE.DAT FILE (cont.)

(i) = Integer number (r) = Real number

Variable **Description**

LFAILGRID(LF) - (i) The floodplain grid element number with a levee that may potentially fail. See comment 1.

LGRIDNO(L) - (i) The grid element number containing the levee segment (see comment 2).

RAISELEV - (r) Incremental height (ft or m) that all the levee grid element crest elevations are raised.

Instructional Comments for the LEVEE.DAT File

1. The general operating rules for the levee failure are as follows:

   a. If the levee fails when overtopped by the flow, set FAILEVEL and FAILTIME = 0.

   b. If the levee is to fail at specified level below the top of the levee, set FAILEVEL equal to the failure elevation.

   c. If the levee is to fail at a specific level below the top of the levee after being inundation for a specific duration, assign appropriate values to both the FAILEVEL and FAILTIME variables.

   d. If the levee fails to a new base levee that is different than the floodplain elevation, assign an appropriate elevation to LEVBASE.

   e. After levee failure, flows will occur through a breach width of FAILWIDTH for that flow direction.

   f. 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.

2. No multiple channels will be assigned to grid elements with levees. Multiple channels in a levee grid element are eliminated automatically by the model.

3. Each levee grid element can have up to eight failure directions. Note that the failure with will be based on an octagon with is 0.4142 times the side length of the grid.

4. Flow over a levee is computed as broadcrested weir flow using a coefficient of 2.85 until the tailwater depth is 85% of the headwater depth. The flow then reverts to overland flow based on the water surface elevations on each side of the levee and the flow depth over the levee.

V. DATA ARRAY SIZE LIMITATIONS AND VARIABLE RANGE

This section is a quick reference for the limits of the variable arrays. Almost all of the arrays are allocatable in the FLO-2D model. In other words, the arrays are defined at runtime. The variables listed below represent the only limitations in array size and will probably never be exceeded for any flood simulation. Arrays are indicated with an I or J index in parentheses. The reference list below is organized in alphabetical order.

Variable	Limitations
NNXSEC(I)	Number of floodplain elements in a cross section, limit = 1000.
NPCHO	Number of channel outflow hydrograph sets, limit = 50.
NPSTA	Number of pairs of channel stage-time entries, limit = 50.
R(I)	Number of discretized rainfall distribution pairs, limit = 1000.

The following attributes have no variable name, but are limited in the array allocation.

Number of discretized inflow hydrograph pairs, limit = 1000.
Number of discretized floodplain inflow hydrograph pairs, limit = 1000.
The number of channel confluences is limited to 100 or less.

The following is a list of variables and their practical range organized by file and by variable name as it appears in the Input Data file variable listing by line. Some of the switches with a range of 0,1,2, etc. are actually check boxes in the Windows or have alpha-numeric text selections in the boxes.

Variable Name	Description	Range
CONT.DAT FILE		
SIMUL	Simulation Time	0.01 - ∞
TOUT	Output Interval	0.01 - ∞
LGPLOT	Graphics Display	0 or 2
METRIC	Metric Switch	0 or 1
IBACKUP	Backup File Switch	0 or 1
ICHANNEL	Channel Switch	0 or 1
MSTREET	Streets Switch	0 or 1
LEVEE	Levee Switch	0 or 1
IWRFS	Reduced Area Nodes	0. - 1.0
IMULTC	Rills and Gullies Switch	0 or 1
IRAIN	Rainfall Switch	0 or 1
INFIL	Infiltration Switch	0 or 1
IEVAP	Evaporation Switch	0 or 1
MUD	Mud/Debris Switch	0 or 1
ISED	Sediment Transport Switch	0 or 1
IHYDRSTRUCT	Hydraulic Structures Switch	0 or 1
IFLOODWAY	Floodway Switch	0 or 1
IDEBRV	Debris Basin Switch	0 or 1
AMANN	n–value	0 - 0.4
XCONC	Concentration	0 – 0.50

XARF	Node Area Reduction	0 - 1.0
FROUDL	Maximum Froude	0 - 5.
SHALLOWN	Shallow Flow n-value	0 - 0.4
ENCROACH	Encroachment Depth	0. – 10.0
NOPRTF	Floodplain Print Options	0, 1, 2 or 3
SUPER	Supercritical Flow Print Option	0 or 1
NOPRTC	Channel Print Options	0, 1 or 2
GRAPTIM	Update Time Intervals	0.1 – 48. (days)
INPLOT	Hydrograph Screen Plot Options	0, 1 or 2

TOLER.DAT

KMODEL	Momentum Equation Selection	2 or 3
TOL	Surface Detention	0.0 - 0.5
DEPTOL	Maximum Depth Increment	0.0 - 0.5
WAVECEL	Wave Celerity Weighting Factor	0.01 - 10.
WAVEMIN	Minimum Stability Coefficient	0.01 - 100.
WAVEMAX	Maximum Stability Coefficient	0.05 - 100.
ITIMTEP	Time Lapse Output Switch	0 or 1
TIMTEP	Time Lapse Output Interval	0.0 - 100.
DTMIN	Minimum Timestep	0.01 – 10.
DTMAX	Maximum Timestep	30.0 - 100.

FPLAIN.DAT

DUM	Grid Element Number	1 - NNOD
FP(I,J); J=1,4	Neighbor Element in Direction N, E, S, W	1 - NNOD
FP(I,5)	n-value	0.010 - 0.40
FP(I,6)	Elevation	0. - 30,000.

INFLOW.DAT

NOHYDRO	Number of Hydrographs	1 – 10000
IHOURDAILY	Hour or Daily Switch	0 or 1
IDEPLT	Hydrograph Plot Node	1 – NNOD
IFC	Node Line Character	'C' or 'F'
INOUTFC	Inflow or Outflow Hydrograph Switch	0 or 1
KHIN	Node	1 – NNOD
HYDCHAR	Hydrograph Line Character	'H'
Time Step	IFPTIME	0 or 1
Hydrograph Entries	NPFPI	1 - 1000
HP(I,J,1)	Time	0.0 - ∞
HP(I,J,2)	Discharge	0.0 - ∞
HP(I,J,3)	Sediment Concentration	0. - 1.

OUTFLOW.DAT

NDC	Number of Outflow Nodes	1 - ∞
NODDC	Outflow Node	1 - NDC
NCHO	Number of Channel Outflow Elements	1 - ∞
Number of Outflow Elements	NCHO	1 - NNOD
KOUT	Outflow Channel Element	1 - NNOD
NPCHO	Number of Stage-Q Outflow Entries	1 - 50

HOUT(N,J,1)	Outflow Rating Depth	0.01 - ∞
HOUT(N,J,2)	Outflow Discharge Coefficient	0.0 - ∞
HOUT(N,J,3)	Outflow Discharge Exponent	0.0 - ∞
NXSEC	Cross Section Number of Nodes	1 - ∞
NXPRT	Cross Section Print Summary Option	0 or 1
IFLO	Cross Section Flow Direction	1 - 8 (-)
NNXSEC	Number of Cross Section Nodes	1 - 1000
NODX	Cross Section Grid Element	1 - NNOD
NSTA	Number of Stage/Time Nodes	1 - NNOD
NOSTA	Stage/Time Grid Element	1 - NNOD
NOSTACFP	Node Switch (Floodplain or Channel)	0 or 1
NPSTA	Number of Stage Entries	1 - 50
STA_TIME(N,J,1)	Time	0. - ∞
STA_STAGE(N,J,2)	Elevation	0. - ∞
DEPTHDUR	Floodplain flow duration depth	0.01 - 100.

RAIN.DAT

IRAINREAL	Real-time Rainfall Switch	0 or 1
RTT	Total Storm Rainfall Depth	0. - ∞
RAINABS	Rainfall Abstract	0. - 1.
IRAINARF	Rainfall Depth-Area Reduction Switch	0 or 1
MOVINGSTORM	Moving Storm Switch	0 or 1
RAINCHAR	Rainfall Line Character	'R'
R_TIME(I1)	Rainfall Time	0. - ∞
R_DISTRIB(I)	Percent Total Rain	0. - 1.0
RAINSPEED	Storm Speed	0. - 100.
IRAINDIR	Storm Direction	1 - 8
IRGRID(I)	Depth Area Reduction Element	1 - NNOD
RAINARF	Rainfall Depth Area Reduction	0. - 1.

INFIL.DAT

ABSTR	Initial Abstraction	0. - 1.
SATI	Soil Initial Saturation	0. - 0.95
SATF	Final Saturation	0.5 - 1.0
POROS	Porosity	0.3 - 0.5
INFCHAN	Channel Infiltration Switch	0 or 1
HYDCALL	Global Hydraulic Conductivity	0.01 - 10.
SOILALL	Global Capillary Suction	1. - 20.
INFILCHAR	Floodplain Infiltration Line Character	'F'
INFGRID	Individual Floodplain Infiltration Node	1 - NNOD
HYDC	Individual Floodplain Hydr. Conductivity	0.01 - 10.
SOILS	Individual Floodplain Capillary Suction	1. - 20.
DTHETA	Individual Node Soil Moisture Deficiency	0. - 0.5
ABSTRINF	Individual Floodplain Node Abstraction	0. - 1.0
RTIMPF	Individual Node Impervious Area	0. - 1.0
HYDCHN	Global Channel Hydraulic Conductivity	0.01 - 10.
INFCHAR	Channel Infiltration Line Character	'C'
INFCH	Individual Channel Infiltration Node	1 - NNOD
HYDCONCH	Individual Channel Hydr. Conductivity	0.01 - 10.

EVAPOR.DAT

IEVAPMONTH	Starting Month of Evaporation	1 - 12
IDAY	Evaporation Starting Day of the Week	1 - 7
ICLOCKTIME	Evaporation Starting Time of the Day	0. - 24.
EMONTH	Name of the Month	(Jan - Dec)
EVAP	Monthly Evaporation Rate	0. - 100.
EVAPER	Percent of Daily Evaporation	0. - 1.0

CHAN.DAT

ICHDEPT	Bed Elevation Switch	0, 1 or 2
DEPINITIAL	Initial Depth	0. - ∞ (-1)
FROUDC	Channel Limiting Froude Number	0. - 5.0
ROUGHADJ	Depth n-value Adjustment Coefficient	0. - 1.2
ISED	Channel Sediment Transport Switch	0 or 1
SHAPE	Channel Geometry Character	'R', 'V', 'T' or 'N'
ICHANGRID	Channel Element Number	1 - NNOD
IQDIR	Channel Extend Direction	1 - 8
BANKELL	Left Bank Elevation	0.01 - ∞
BANKELR	Right Bank Elevation	0.01 - ∞
FCN	Channel n-value	0.01 - 0.25
FCW	Channel Width	0.1 - ∞
FCD	Channel Depth	0.01 - 1000.
XLEN	Channel Length	0.01 - ∞
A1, A11	Channel Regression Area Exponent	0. - ∞
A2, A22	Channel Regression Area Coefficient	0. - ∞
B1, B11	Channel Regression Wetted Perimeter Exp.	0. - ∞
B2, B22	Channel Regression Wetted Perimeter Coef.	0. - ∞
C1, C11	Channel Regression Top Width Exp.	0. - ∞
C2, C22	Channel Regression Top Width Coef.	0. - ∞
ZL	Trapezoidal Geometry, Left Slope	0.01 - 100.
ZR	Trapezoidal Geometry, Right Slope	0.01 - 100.
NXSECNUM	Channel Cross Section Number	1 - NNOD
ISTART	Starting Water Surface Elevation Node	1 - NNOD
WSELSTART	Starting Water Surface Elevation	0.01 - 30,000.
IEND	Last Node with Starting WS Elevation	0 - NNOD
WSELEND	Ending Water Surface Elevation	0.01 - 30,000.
NOFLO1	NOFLOC Channel Element	1 - NNOD
NOFLO2	NOFLOC Channel Element	1 - NNOD

XSEC.DAT

XSECHAR	Cross Section Line Character	'X'
XSECNUM	Cross Section Number	1 - NNOD
XSECNAME	Cross Section Name	Alpha-numeric name
XI	Cross Section Station	0. - ∞
YI	Cross Section Elevation	0. - 30,000.

HYSTRUC.DAT

STRUCHAR	Hydraulic Structure Line Character	'S', 'C', 'R' or 'T'
STRUCTNAME	Hydraulic Structure Name	Alpha-numeric name
IFPORCHAN	Floodplain or Channel Switch	0 or 1

ICURVTABLE	Rating Curve or Table Switch	0 or 1
INFLONOD	Structure Inflow Node	1 - NNOD
OUTFLONOD	Structure Outflow Node	1 - NNOD
INOUTCONT	Inflow or Outflow Control Switch	0 or 1
HEADREFEL	Structure Headwater Reference Elevation	0.01 - 30,000.
HDEPEXC	Structure Rating Curve Reference Depth	0.01 - 1000.
COEFQ	Structure Rating Curve Coefficient	0. - ∞
EXPQ	Structure Rating Curve Exponent	0. - ∞
REPDEP	Replacement Rating Curve Depth	0.01 - ∞
RQCOEF	Replacement Rating Curve Coefficient	0. - ∞
RQEXP	Replacement Rating Curve Exponent	0. - ∞
HDEPTH	Structure Rating Table Depth	0.01- 1000.
QTABLE	Structure Rating Table Discharge	0.01 - ∞

STREET.DAT

STRMAN	Global n-value for Street Flow	0.01 - 0.25
ISTRFLO	Switch, Inflow Hydrograph Enter Street	0 or 1
STRFNO	Limiting Street Froude Number	0. - 5.
DEPX	Street Curb Height	0.01 - 2.
WIDST	Global Street Width	0.01 - ∞
STCHAR	Street Line Character	'N', 'S', or 'W'
STRNAM	Street Name	Alpha-numeric name
IGRIDN	Street Grid Element	1 - NNOD
DEPEX	Curb Height	0.0 - 2.
STMAN	Street n-value	0.01 - 0.25
ELSTR	Street Elevation	0. - 30,000.
ISTDIR	Street Flow Direction	1 - 8
WIDR	Street Width	0. – 1000.

ARF.DAT

ITTCHAR	ARF Line Character	'T'
ITTAWF	Blocked Node	1 - NNOD
IGD	ARF Grid Element	1 - NNOD
ARF	Area Reduction	0. - 1.0
WRF	Width Reduction Factor	0. - 1.0

MULT.DAT

WMC	Multiple Channel Incremental Width	0. - 1000.
WDRALL	Global Multiple Channel Width	0. - 1000.
DMALL	Global Multiple Channel Depth	0. - 1000.
NODCHNSALL	Global Multiple Channels per Element	1 – 100
XNMULTALL	Global Multiple Channel n-value	0.01 - 0.5
IGRID	Multiple Channel Element	1 - NNOD
WDR	Grid Element Multiple Channel Width	0. - 1000.
DM	Grid Element Multiple Channel Depth	0. - 1000.
NDCHN	Grid Element Number of Multiple Channels	0 - 100
XNMULT	Grid Element Multiple Channel n-value	0.01 - 0.5

SED.DAT

VA	Viscosity vs. Sediment Conc. Coefficient	0. - ∞
VB	Viscosity vs. Sediment Conc. Exponent	0. - ∞
YSA	Yield Stress vs. Sediment Conc. Coefficient	0. - ∞
YSB	Yield Stress vs. Sediment Conc. Exponent	0. - ∞
SGS	Sediment Specific Gravity	2.5 - 2.8
XKX	Laminar Flow Resistance	24 - 50,000
ISEDEQ	Sediment Transport Equation Identifier	1 - 6
DFIFTY	Sediment Median Size D_{50}	0.0625 - ∞
SGRAD	Sediment Gradation Coefficient	1. - 10.
SGST	Sediment Specific Gravity	2.6 - 2.8
DRYSPWT	Sediment Dry Specific Weight	70. - 130.
ISEDSUPPLY	Sediment Supply Rating Curve Switch	0 or 1
ISEDISPLAY	Sediment Transport Capacity Grid Element	1 - NNOD
JDEBNOD	Grid element with Debris Basin	1 - NNOD
DEBRISV	Debris Basin Volume	0. - ∞
SCOURDEP	Maximum Allowable Scour Depth	0. - 100.
SEDCHAR	Sediment Line Character	'R' or 'S'
ICRETN	Grid Elements with a Rigid Bed	1 - NNOD
ISEDGRID	Sediment Supply Rating Curve Element	1 - NNOD
ISEDCFP	Sediment Supply Element Identifier	0 or 1
ASED	Sediment Supply Rating Curve Coef.	0. - ∞
BSED	Sediment Supply Rating Curve Exp.	0. - ∞

LEVEE.DAT

RAISELEV	Incremental Increase in Levee Crest	0. - 100.
LEVCHAR	Levee Line Character	'L', 'D', 'F' and 'W'
LGRIDNO	Levee Grid Element	1 - NNOD
LDIR	Levee Cutoff Flow Direction	1 - 8
LEVCREST	Levee Crest Elevation	0.01 - 30,000.
LFAILGRID	Levee Element that may Fail	1 - NNOD
LFAILDIR	Potential Levee Failure Direction	1 – 8
FAILELEV	Levee Failure Elevation	0.01 - LEVCREST
FAILTIME	Levee Failure Time Duration	0. - SIMUL
LEVBASE	Levee Failure Base Elevation	0.01 - LEVCREST
FAILWIDTH	Levee Fail Width	0.01 - SIDE*0.4142
FAILRATE	Levee Vertical Fail Rate	0.01 - 1000.
FAILWIDRATE	Width Fail Rate	0.01 - 1000.

VI. OUTPUT FILES AND OPTIONS

During the flood simulation, the user has a choice of two screens to watch, output text screen and a graphical simulation of the floodwave progression. 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. This screen enables the user to view the simulation runtime speed and the numerical stability of the run. The second screen is a color graphics display of the grid element maximum flow depth on the entire system along with the inflow hydrograph. After the flood simulation is complete, the hydraulic results are presented in a series of output files that contain hydrographs, maximum flow depths and maximum velocities.

A FLO-2D project application may consist of a series of flood scenarios. The conservation of flood volume (inflow equals outflow plus storage) should be reviewed with each simulation. A summary of the inflow volume, final volumes left on the floodplain (storage) and outflow from the grid system is presented in the BASE.OUT and SUMMARY.OUT files. If the inflow does not equal the final storage on the grid system plus outflow from the grid system within a fraction of one percent, there is problem with the simulation.

Options are available to create output files with either a spatial or temporal format. Output files that have substantial detail involving all the timesteps or all the grid elements, can be generated. Some output files are designed for a simple review of specific flow hydraulics. Output data include water surface elevation, flow depth, velocity, discharge and sediment concentration. Overland flow hydraulics may be viewed as individual grid elements or the grid elements can be grouped together to produce floodplain cross sections. Summary tables listing maximum velocity and flow depths and their times of occurrence appear near the end of the BASE.OUT file.

The user is cautioned about specifying the output options 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. For example, if an output interval (TOUT) of 0.1 hours is selected for a 5-hour flood event, 50 sets of all the computed flow hydraulics for every grid element will be written to the BASE.OUT file. If TOUT equals 0.01 hours for a 3000 element system and a 50 hour storm, over 15 million lines of data would be written in the BASE.OUT file. Setting NOPRTFP = 2 will turn off all of the temporal floodplain hydraulic output data.

The user does not have to specify any output file names. Simply clicking on the FLO.EXE in the GUI executable menu or double clicking on FLO.EXE in Explorer runs the model and produces the output files. ***It is important to note that each time FLO-2D is run, it will write over the existing output files and the previous output file data will be lost. To save any output files in anticipation of subsequent FLO-2D runs, the user must rename or move them to another subdirectory.***

Use WordPAD or any other ASCII editor to view the output files. A brief description of all the output files follows:

BASE.OUT

BASE.OUT is the primary output file. At the beginning of the file, the inflow hydrographs are printed, then the time dependent output data follows. For each specified time output interval, the flow depth, velocity, water surface elevation and discharge for either the channel or the floodplain grid elements can be written. The user will probably have little interest in the floodplain hydraulics for the individual grid elements and this portion of the file should be turned off for most applications (set NOPRTFP = 2). The outflow from the boundary grid elements is listed at the end of the time interval. After the final time output interval, a summary of all the grid elements maximum depths, water surface elevations, velocities and the time of occurrence of the maximum values is printed. Finally, a summary table of the inflow, outflow and storage volumes at the end of the file allows the user to review the conservation of mass and the ultimate disposition of all the water and sediment. For convenience, this conservation table is also written to a separate file named SUMMARY.OUT. There is so much information in the BASE.OUT file that the user is encouraged to review all the various data output options before starting a flood simulation.

CHANBANKEL.CHK

When the bank elevations prescribed in the CHAN.DAT do not match the corresponding floodplain grid element elevations, the difference in elevation is reported in this file. If the difference between the channel bank elevation and the floodplain grid element elevation exceeds 1 ft (0.3 m), the model reports it in this file. This includes the channel extension grid elements. If the 1 ft (0.3 m) limit is exceeded during the simulation, the model assumes that the bank elevation is more accurate and resets the floodplain elevation to the bank elevation.

CHANMAX.OUT

The maximum discharge and stage for each channel element and the corresponding time of occurrence is printed to this file. For mudflows, this file will also list the maximum sediment concentration and the time of occurrence.

CHANNEL.CHK

When the channel cross section width exceeds the grid element width, the cross section extends into neighboring elements. This file provides a list of all channel extensions that are required by a cross section.

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 channel water surface elevation.

CHNBEDEL.OUT

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

CHVOLUME.OUT

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

CONFLUENCE.OUT

This file lists the channel elements that constitute a confluence as defined by having three or more channel elements contiguous to a given channel element. The confluence may be a prescribed condition or it could constitute a data error in the CHAN.DAT requiring NOFLOC definition.

CROSSMAX.OUT

When the floodplain cross section analysis is requested in the FPINOUT.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 element grid hydrographs for each of the floodplain elements in the cross section. The time and discharge are listed for each output interval.

DEPTH.OUT and Related 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 drawing to create maximum flood depth contours on the digitized base map. The following output files are created:

CHNBEDEL.OUT - Channel bed elevations;
DEPCH.OUT - Maximum channel flow depths;
DEPFP.OUT - Maximum floodplain flow depths;
DEPTH.OUT - Maximum combined channel and floodplain flow depths;
FINALDEP.OUT - Final floodplain flow depths;
STRELEV.OUT - Street elevations.

Two output files are created with the following format for Intergraph usage: x- and y-coordinates, maximum flow depth or elevation. No grid element numbers are included.

DEPTHTOL.OUT - Maximum combined channel and floodplain flow depths greater than the TOL value. Values less than the TOL value are set to zero.

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.

DEPTHDUR.OUT

DEPTHDUR.OUT contains the floodplain inundation duration data including the grid element number, x- and y- grid element 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.

ERROR.CHK

The ERROR.CHK file contains input data error comments when the program terminates prior to the hydraulic computations. Some input data errors are recognized at execution time by the computer and are not listed in this file. The backup data files (*.BAC) should be reviewed with this file to determine if the input data is being read properly at runtime.

HYCHAN.OUT

This channel hydraulics output file contains a hydrograph for each channel element and includes the time, elevation, depth, velocity, discharge and sediment concentration. The maximum discharge and stage are also listed with their times of occurrence. Additional channel flow hydraulic and geometry parameters include: flow area, hydraulic radius, wetted perimeter, top width, width-to-depth ratio, energy slope, and bed shear stress.

HYCROSS.OUT

The output interval time, top width, depth, velocity and discharge are listed in the file for each cross section. The data shows the discharge passing the cross section of grid elements in the form of a hydrograph. The maximum discharge for the cross section and the individual grid elements is printed in the CROSSMAX.OUT file.

HYDROSTRUCT.OUT

The discharge hydrograph from any hydraulic structure is presented in this output file.

IMPACT.OUT

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

INFILHY.OUT

The hydraulic conductivities are listed in this file to view the spatial variation. This file contains grid element number, coordinates and floodplain hydraulic conductivity.

INTERGWS.OUT

Maximum floodplain water surface elevations based on the grid element elevation in the TOPO.ORI. Values less than TOL are set to zero.

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.

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 size will enlarge to contain the flow. This occurs when the flow depth exceeds the specified channel depth and the gully 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:

File: MULTCHN.OUT

MULTIPLE CHANNEL WIDTH REVISIONS

NODE	MAX WIDTH	FINAL WIDTH	ORIG WIDTH
2	425.	384.	50.
28	114.	20.	20.
29	169.	89.	20.
31	15.	5.	5.
55	218.	13.	10.
56	181.	10.	10.
57	92.	10.	10.
58	41.	10.	10.
59	50.	5.	5.
85	437.	117.	5.
86	280.	5.	5.
87	120.	5.	5.
88	137.	5.	5.
89	132.	5.	5.
113	204.	3.	3.

OUTNQ.OUT

The OUTNQ.OUT file contains a summary of the maximum discharge, time of peak and the discharge hydrograph for each floodplain outflow element.

OVERBANK.OUT

When the flow exceeds bankfull discharge and begins to inundate the floodplain, the channel grid element and time of overbank flood occurrence are written to this file.

ROUGH.OUT

The ROUGH.OUT file tracks the changes in the Manning's n roughness values. The user specifies a maximum Froude number for overland, channel and street flow. When the computed Froude number exceeds the specified maximum value for a given grid element, the n-value for that grid element is increased by 0.001. During the falling limb of the hydrograph when the Froude number drops below 0.5, the n-value is decreased by 0.0005 until the original n-value is reached. The maximum, final and original n-values for floodplain, channel and street are listed in ROUGH.OUT by grid element. A portion of a ROUGH.OUT file follows:

FLOODPLAIN MANNINGS N CHANGES
FP = FLOODPLAIN, MC = MULTIPLE CHANNELS, ST = STREETS

	NODE	MAX n	FINAL n	ORIG. n
FP	1171	0.067	0.067	0.065
FP	1172	0.069	0.069	0.065
FP	1250	0.072	0.072	0.065
FP	1251	0.076	0.074	0.065
FP	1252	0.079	0.077	0.065
FP	1253	0.103	0.103	0.065
FP	1254	0.079	0.077	0.065
FP	1255	0.079	0.077	0.065
FP	1331	0.078	0.076	0.065

CHANNEL MANNINGS N CHANGES

NODE	MAX n	FINAL n	ORIG. n
2998	0.085	0.054	0.050
2985	0.083	0.070	0.050
2968	0.083	0.062	0.050
2946	0.101	0.066	0.050
2922	0.087	0.074	0.050
2894	0.106	0.074	0.050
2863	0.066	0.064	0.050

SEDCHAN.OUT

The sediment transport routine will compute scour and deposition in the channel. In this output file, the x- and y-coordinates, maximum deposition, maximum scour and final bed elevation change (ft or m) are reported by channel element. With the x- and y-coordinates for the grid elements, the user has the opportunity to plot the data in a CADD drawing.

SEDFP.OUT

Similar to the SEDCHAN.OUT file, the floodplain scour and deposition are reported in the SEDFP.OUT file. The x- and y-coordinates, maximum deposition, maximum scour and final bed elevation change (ft or m) for the floodplain surface are reported by grid element.

SEDTRAN.OUT

The sediment transport capacity (cfs or cms) computations for each of the six sediment transport equations are listed by output interval in this file.

SPECENERGY.OUT

The specific energy is the sum of the depth plus the velocity head. This file lists the maximum specific energy (ft or m) for a floodplain grid element and includes grid element number, x- and y- grid element coordinates and maximum specific energy.

STATICPRESS.OUT

The spatially variable static force per linear foot for each floodplain element is presented is this file by grid element number, x- and y-coordinates and force per linear foot.

STREET.OUT

Similar to DEPTH.OUT, this file contains the grid element x- and y-coordinates and the maximum street flow depth for each grid element with at least one street. This file can be imported to the CADD drawing to create flood depth contours on the digitized base map.

SUMMARY.OUT

This file is a replication of the text screen output and it includes the volume conservation summary table at the end of the BASE.OUT file. It contains a list of the simulation output time interval, the minimum timestep and the volume conservation and reports the inflow hydrograph volume, rainfall volume, infiltration loss volume, outflow and storage volumes. This file can be reviewed to determine if and when the volume conservation was violated and the final distribution of volume in the system.

SUPER.OUT

When the floodplain or channel flow is supercritical, the grid elements experiencing supercritical flow, Froude number, flows and time of occurrence are written to this file. This file requires that the variable SUPER = 1 in the CONT.DAT file.

SURFAREA.OUT

The SURFAREA.OUT lists the available flow surface in each grid element. The area reduction factors (ARF) remove a portion of the surface area 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 overland flow area is reported. At the end of the file, the maximum area of floodplain inundation (including the channel surface area) for the entire grid system is listed by output time interval. This can be an informative data file for the user. If the available flow surface area is less than 5% of the original grid element surface, the FLO-2D simulation will be terminated. The SURFAREA.OUT file will permit a review of how the surface area is divided up between the various flow components. The first part of the SURFAREA.OUT file has the following format:

TABLE OF GRID ELEMENT SURFACE AREAS

UNMODIFIED GRID ELEMENT SURFACE AREA (SQ. FT) = 250000.

NODE	ARF-REDUCED AREA	CHANNEL AREA	STREET AREA	MULT CHANNEL AREA	OVERLAND AREA	MULT CHANNELS (0-NO, 1-YES)
1	175000.	0.	0.	0.	175000.	0
2	175000.	0.	0.	0.	175000.	0
3	175000.	109546.	0.	0.	98318.	0
4	175000.	0.	0.	0.	175000.	0
5	175000.	0.	0.	0.	175000.	0
6	175000.	125790.	0.	0.	86947.	0
7	175000.	0.	0.	0.	175000.	0
8	175000.	0.	0.	0.	175000.	0
9	175000.	0.	0.	0.	175000.	0
10	175000.	167509.	0.	0.	57744.	0
11	175000.	0.	0.	0.	175000.	0

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. If a 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 review. This file has the following format:

TIMESTEPS WERE DECREASED FOR THE FOLLOWING NODES:

GRID ELEMENT	NUMBER OF TIMESTEP DECREMENTS
FLOODPLAIN NODES	
2666	63.
1253	19.
1171	12.
2710	11.
2512	9.
2568	7.
2456	6.
2122	5.
2457	5.
2513	5.
1413	4.
1251	3.
1334	3.
CHANNEL NODES	
2946	169.
1655	51.
1092	43.
1813	16.
1495	13.

TIMDEP.OUT

This file contains flow depths for each floodplain grid element at the user specified time intervals. The grid element x- and y-coordinates are written in this file so that this data can be imported to the CADD drawing. From this data file a series of contour maps can be created in the CADD drawing to display a time-lapse sequence of the flood progression over the alluvial fan. This file is also required for a time-lapse simulation in the MAXPLOT and MAPPER post-processor programs.

VELOC.OUT and Related Velocity Output Files

These files are similar to the DEPTH.OUT file except that the maximum velocity for each floodplain or channel element is written in place of the maximum depth. These files also contains the x- and y-coordinates so that the data can be imported to the CADD drawing and velocity contours can be plotted. These files can be viewed with the MAXPLOT or MAPPER program. The velocity output files include:

STVEL.OUT - Maximum street flow velocity;
STVELDIR.OUT - Flow direction of the maximum street flow velocity;
VELFP.OUT - Maximum floodplain flow velocity;
VELOC.OUT - Maximum combined channel and floodplain flow velocity;
VELDIREC.OUT - Flow direction of the maximum floodplain flow velocity.
FINALVEL.OUT -Flow velocity at the end of the simulation.
FINALVELDIR.OUT - Flow maximum velocity direction at the end of the simulation.

XSECAREA.OUT

When the channel cross section option is invoked for channel routing, the channel geometry data is written to this file. It includes: grid element, flow area, top width and wetted perimeter for the lowest top of bank (bankfull flow).

XSEC.OUT

This file is created by the sediment transport option (ISED = 1 in CONT.DAT and ISEDN = 1 for a channel segment in CHAN.DAT) for channel cross section data. It contains the final cross section bed elevations after scour and deposition have been computed in a mobile bed channel simulation.

VII. POST-PROCESSOR PROGRAMS

There are four post-processor programs HYDROG, MAPPER, MAXPLOT AND PROFILES that display the output data graphically. Channel flood simulations are required for HYDROG and to view the water surface profiles in the PROFILES post-processor. MAPPER and MAXPLOT display the flood inundation. These files can be initiated by clicking on the name of the program from the Post-Processor pull down menu in the GUI or by copying the executable *.EXE file into the project subdirectory and double clicking on the program name in Explorer.

7.1 HYDROG

Channel output hydrographs can viewed with HYDROG. It displays the hydrograph for every channel element in the system. HYDROG will interactively display the channel hydrograph or list the average channel hydraulic data for the entire system. 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:

Optional HYDRO.DAT file:

- Line 1. Number of gaging stations with hydrograph data.

Lines 2 thru 4 are repeated for each gaging station.

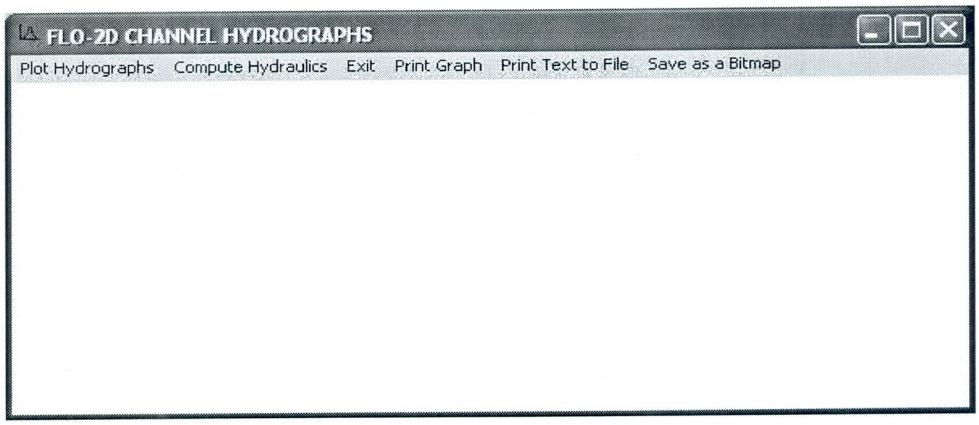
Line 2. Name of the gaging station (limit 10 letters)
Line 3. Channel grid element number and the number of hydrograph data pairs.
Line 4 to End. List the hydrograph pairs in columns, time (hours) and discharge (cfs).

Example HYDRO.DAT file:

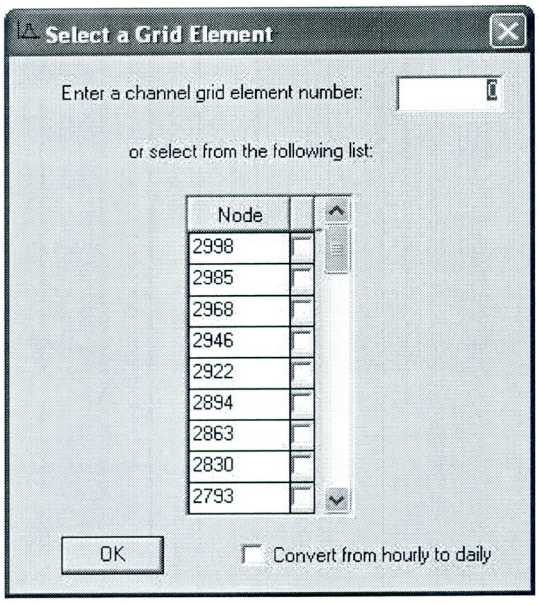
```
      5
  SanFelipe
960         480
0.25   2533.83
0.50   2608.14
0.75   2558.46
1.00   2608.14
1.25   2608.14
1.50   2583.22
1.75   2583.22
2.00   2583.22
2.25   2608.14
2.50   2583.22
2.75   2583.22
3.00   2608.14
3.25   2583.22
  ...
```

Instructional commands for HYDROG follow:

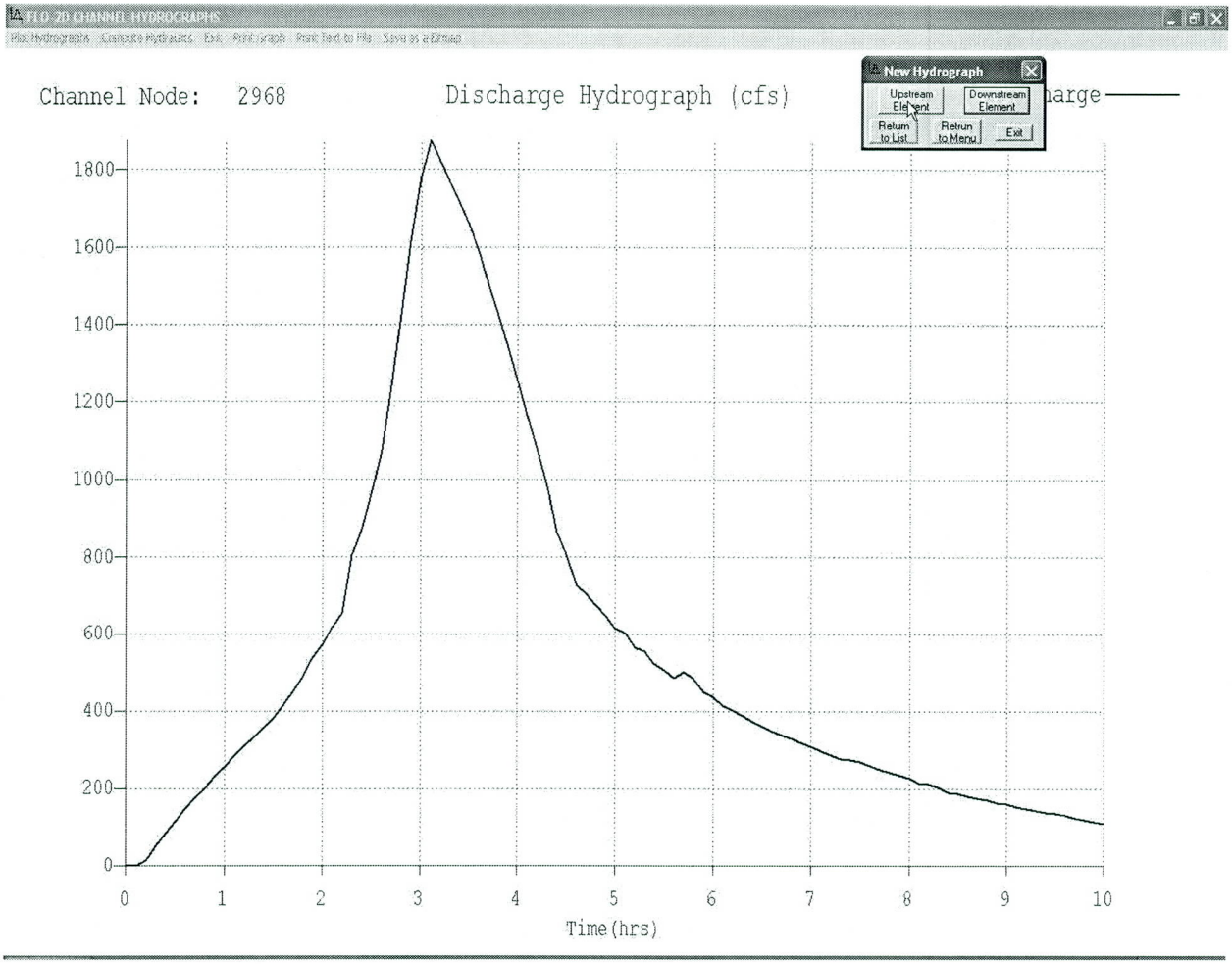
After opening HYDROG, click on either Plot Hydrographs or Compute Hydraulics in the Main Menu shown below:



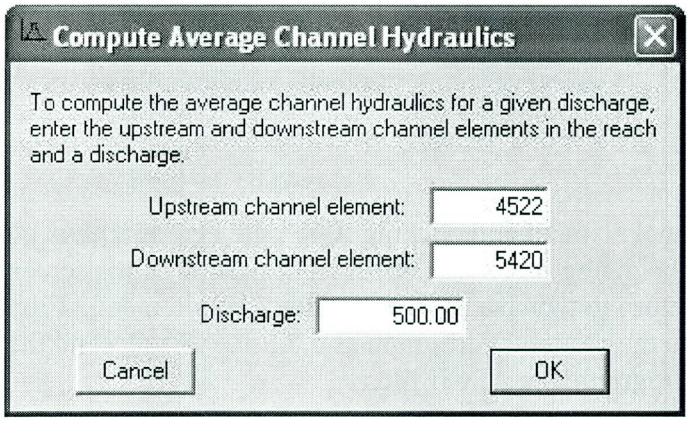
If you click on Plot Hydrographs a dialog box appears so that either a channel segment or element can be selected.



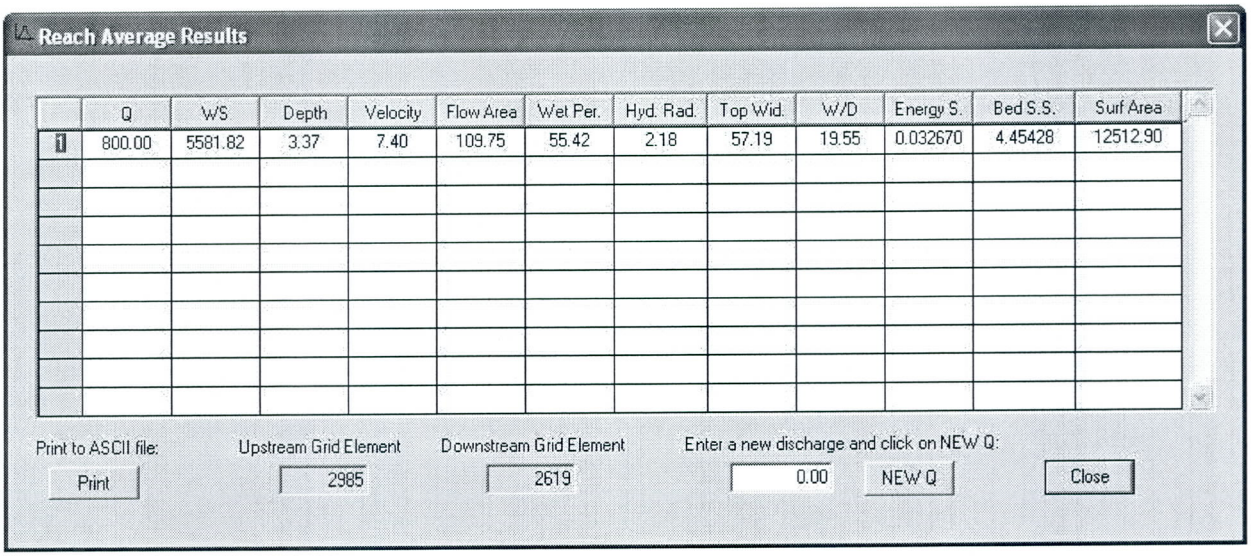
After selecting the channel element and clicking 'OK', the hydrograph is plotted as shown in the following figure. Use the dialog box in the upper right portion of the screen to select another channel element or to return to the channel element list or main menu. From the Main Menu, the other options are to save the plotted hydrograph as a bitmap image, send the hydrograph to a printer, or write the hydrograph to a ASCII file.



If the Compute Hydraulics is selected from the Main Menu, the following dialog box is displayed:



Mouse click 'OK' to display the following table:



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

7.2 MAPPER

FLO-2D Mapper is a post-processor program that generates maps of the results of the *FLO-2D* flood routing model including hydraulic variables, water surface elevations and sediment scour or deposition. Three types of maps can be created: grid element or DTM point plots, contour maps and shaded contour maps. These graphic maps may be created for ground surface elevations, maximum water surface elevations, maximum floodplain flow depths, maximum velocities, velocity vectors, scour and deposition depths, and final bed elevations. This section presents *FLO-2D Mapper* installation instructions and is a guide for the *FLO-2D Mapper* commands.

7.2.1 Introduction

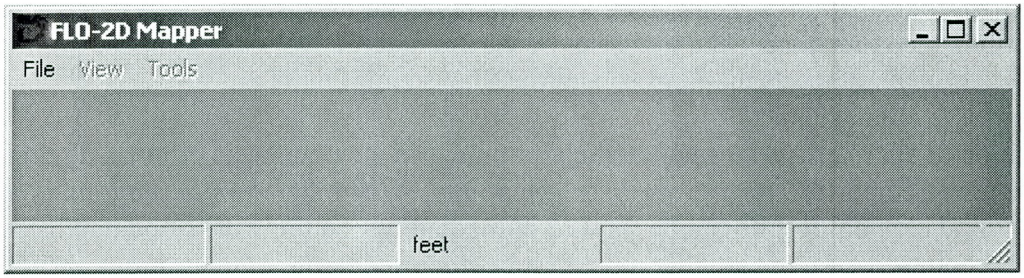
The *Mapper* program provides a graphical tool to view the *FLO-2D* hydraulic results. *Mapper* utilizes the capabilities of the MS-Windows™ operating systems. A Windows 98™ or newer system is recommended. The plots can be viewed interactively using mouse point and click events. *Mapper* supports the English and the SI system of units.

A flood depth map can be generated either on a grid element basis or by analyzing the flow depths above imported DTM ground surface points. When the *Mapper* program is opened, the *FLO-2D* grid element results can be viewed immediately. To view the flood depths over the DTM ground points, a DTM points file must first be imported to the *Mapper* program. The DTM points are then subtracted from the grid element water surface elevation to determine the flow depth over the DTM points. These DTM flow depths can be interpolated with a triangular irregular network to plot the flow depth contours. *FLO-2D Mapper* has the following features:

- The *FLO-2D Mapper* has been completely redesigned to incorporate ESRI MapObjects© GIS software controls.
- ESRI shape file format data like land use, soil types, and Manning roughness coefficients can be imported.
- ESRI ArcInfo ASCII grid files containing terrain elevations can be imported.
- Multiple geo referenced aerial photos in various graphic formats can be imported such as TIFF, BMP, JPG, etc.
- Multiple ESRI ArcInfo ASCII grid files listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- Multiple image files such as aerial photos listed in a tile and index catalog file and referenced to a user defined polygon over the grid system can be imported.
- New multiple layer capability, including control of layer properties is available.
- Improved zoom features and a new pan display have been developed.
- The maximum sediment scour and deposition can be plotted.
- A final bed elevation map can be plotted.
- Flow depths and topography profile cuts can be plotted.
- Velocity vectors are displayed.
- DTM points can be deleted using a polygon to define a area from which the DTM points are deleted.
- A FEMA map template can be imported.

7.2.2. Installing and Starting the MAPPER Program

MAPPER is automatically installed with the FLO-2D model installation. The MAPPER program is placed in the FLO-2D subdirectory and an icon is placed on the desktop. To start the program double-click the MAPPER icon or the MAPPER.EXE in Explorer. You can also start by clicking on the MAPPER name in the GUI Post-Processor pull-down menu. When MAPPER initializes the main window appears:



IMPORTANT NOTE: *For the system to operate properly, the MS-Windows™ environment must be set to a number format with decimal point using the International (Regional) option of the Control Panel.*

7.2.3 Installation of Other Required DLL files

The *Mapper* uses ESRI MapObjects© GIS software controls. These components require several data base libraries that may not exist in all computers. In order to ensure that all required components exist, it is recommended to follow these instructions. ***Please note that these components are also used in the GDS and need only to be installed only once.***

Installation of the other required dynamic link libraries (DLL) files involves the following steps. The GDS uses ESRI MapObjects© GIS software controls. These components require several data base libraries that may not exist in all computers. In order to ensure that all required components exist, follow these instructions.

1. Copy the following files to your computer: MO21rt.EXE and DAO35.exe. You may find these files in the distribution CD.

2. Run these programs with the corresponding arguments as follows:
   MO21rt.EXE /FGHMJ and DAO35.exe /d=ABCDF /ni

2. Restart your computer.

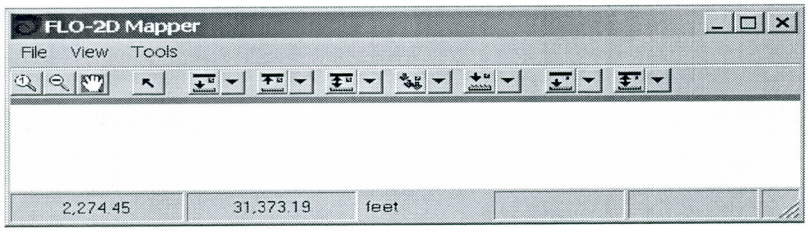
IMPORTANT NOTE*: In some computer systems it will be required to install other Microsoft data DLL component. To verify if you have these DLL components installed, run the MDAC_TYP.EXE program and follow the instructions to register all files. Further directions to apply the MDAC_TYP.EXE can be found at:*

http://www.microsoft.com/data/mdac27info/en/readme.htm

7.2.4 MAPPER Menu and Toolbar Command

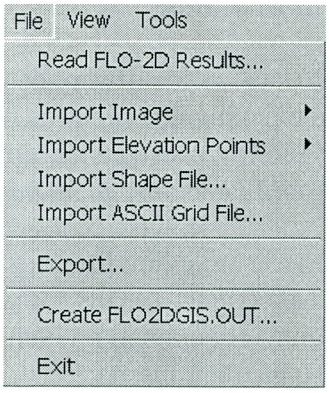
Main Menu

This section describes the available *FLO-2D Mapper* commands in the main menu:

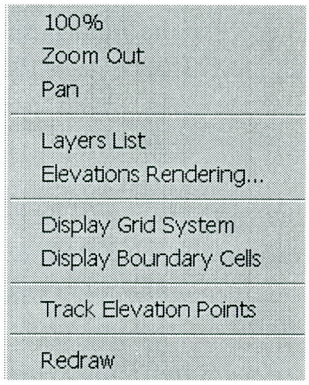


Each menu command may contain a number of subcommands as shown in the following submenus dialog boxes.

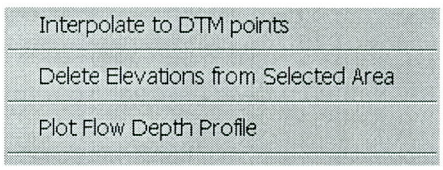
File Commands:



View Commands:



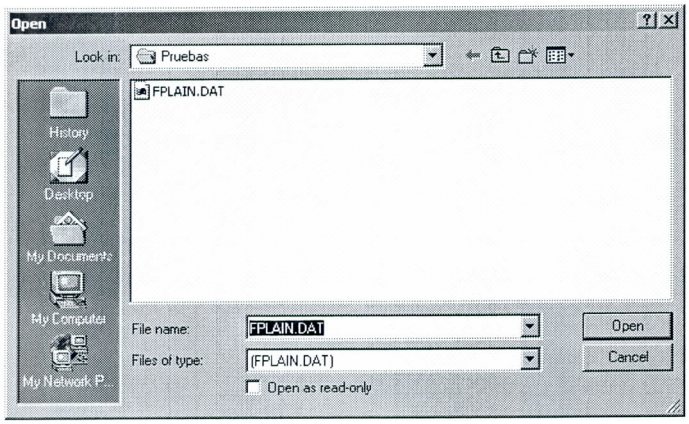
Tools Commands:



File Menu

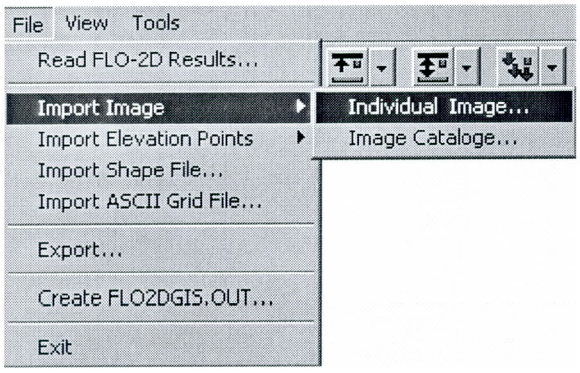
Read *FLO-2D* Results Command… (File Menu)

This command reads data created by the *FLO-2D* model. A dialog box appears to select a FPLAIN.DAT file:



Additional required or complimentary files are automatically read from the same directory.

Import Image/Individual Image (File Menu)



Use this command to import individual images like aerial photos. You may import several images. To correctly position the photo, each photo must be accompanied by a world file containing its georeference data. The world file associated with an image has a name according to the conventions in the table below. For example, if you have an image file named *myimage.bmp*, then the world file associated with it must be named *myimage.bmpw* or *myimage.bpw*

Image File Extension	World File Extension
bmp	bmpw or bpw
jpg; jpeg	jpgw or jgw
tif; tff; tiff	tfw
gis	gsw
lan	lnw
bil	blw
bip	bpw
bsq	bqw
sid	sdw
sun	snw
rs; ras	Rsw
rlc	rcw

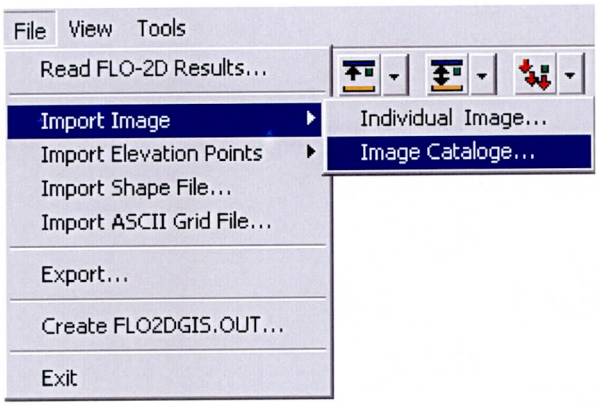
The world file has the following format:

Line1: Dimension of a pixel in map units in the x-direction.
Lines 2 and 3: Not used in this release.
Line 4: This value is always negative because the image space is top-down, whereas map space is bottom-up.
Line 5: Translation term; x-Origin (x-coordinate of the center of the upper left pixel)
Line 6: Translation term; y-Origin (y-coordinate of the center of the upper left pixel)

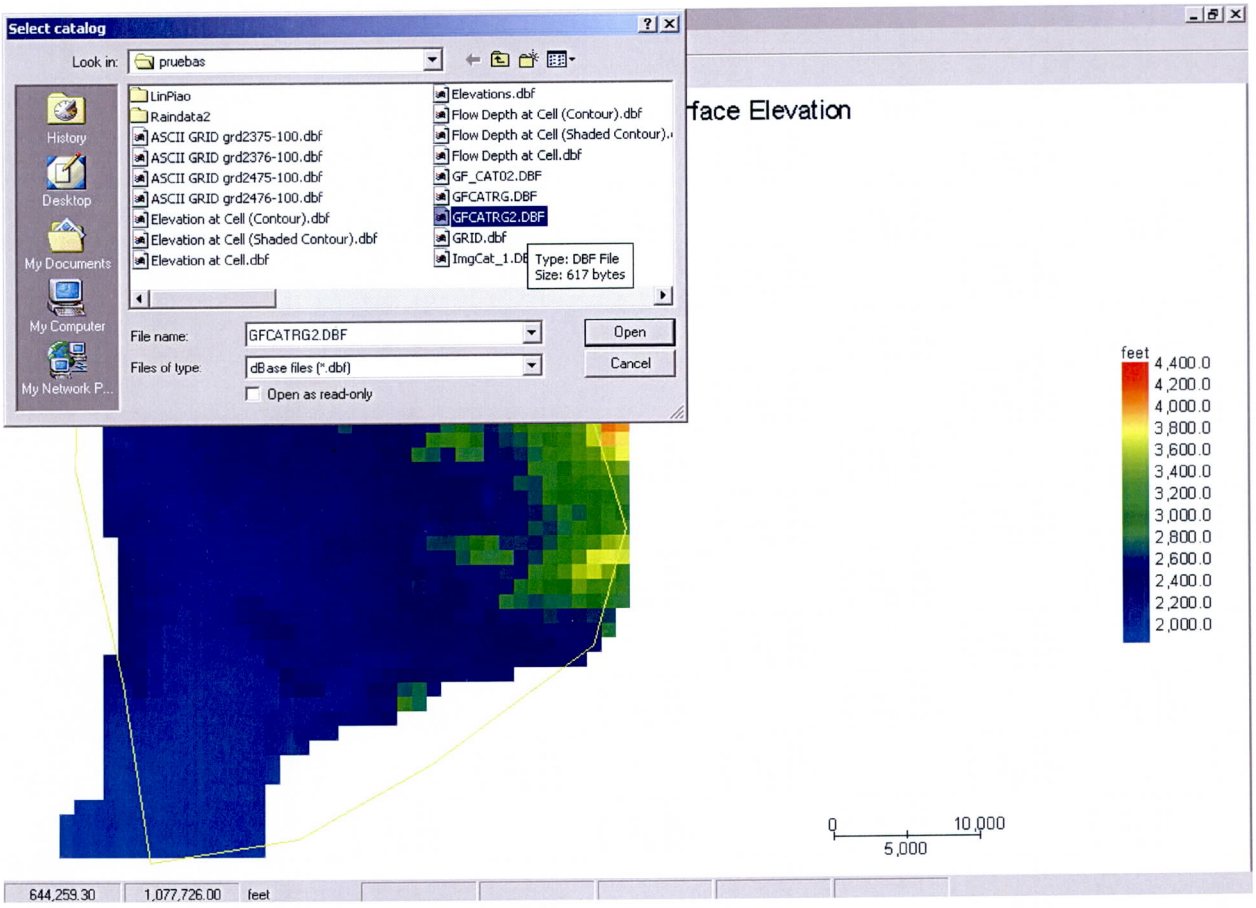
Example:

```
    20
     0
     0
   -20
637510
1032490
```

Import Image/Image Catalogue (File Menu)



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

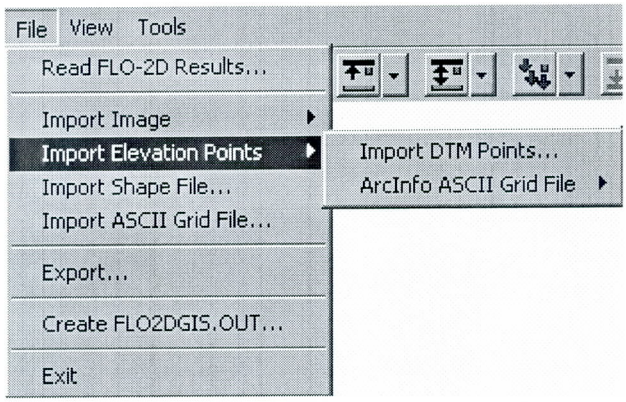


The catalog file may be in DBASE or ASCII format and has the following format:

IMAGE	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\ 6401030-5.TIF	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\ 6401035-5.TIF	660000	1300000	770000	1500000
\\Agua\IMF\Publico IMF\ 6401055-5.TIF	630000	1300000	750000	1510000

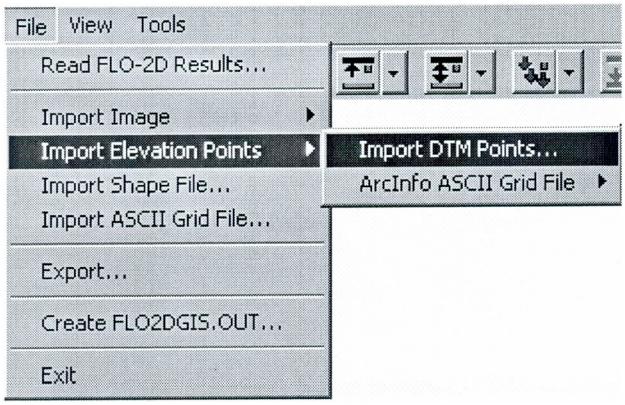
The first column is the file name including its path and the following four columns are the image limits coordinates. *Mapper* will find all images from the catalog that are contained or intersect the user defined polygon and will retrieve the appropriate ones.

Import Elevation Points Command (File Menu)



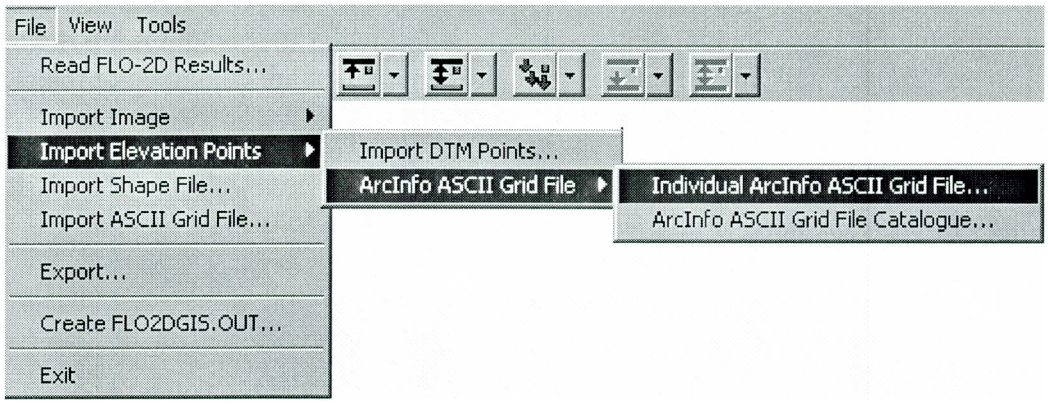
This command will import an existing digital terrain elevation file. There are two submenu commands to import either digital terrain elevation points (DTM Points Command) or ArcInfo ASCII Grid File.

Import DTM Points...Command (File Menu)



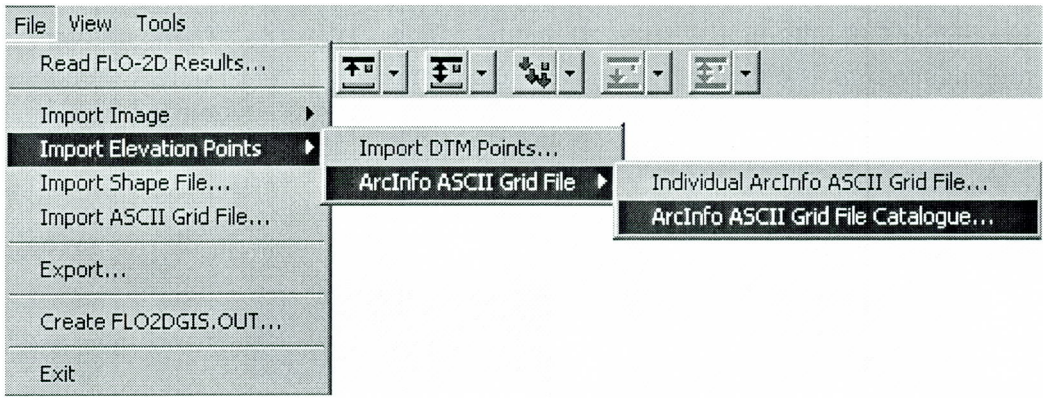
This command imports DTM elevation points from an existing file. You may import several data files. The new points are added to the existing data. You may also mix or combine DTM points with other random elevation points from an ArcInfo ASCII grid files.

Individual ArcInfo ASCII Grid File Command (File Menu)

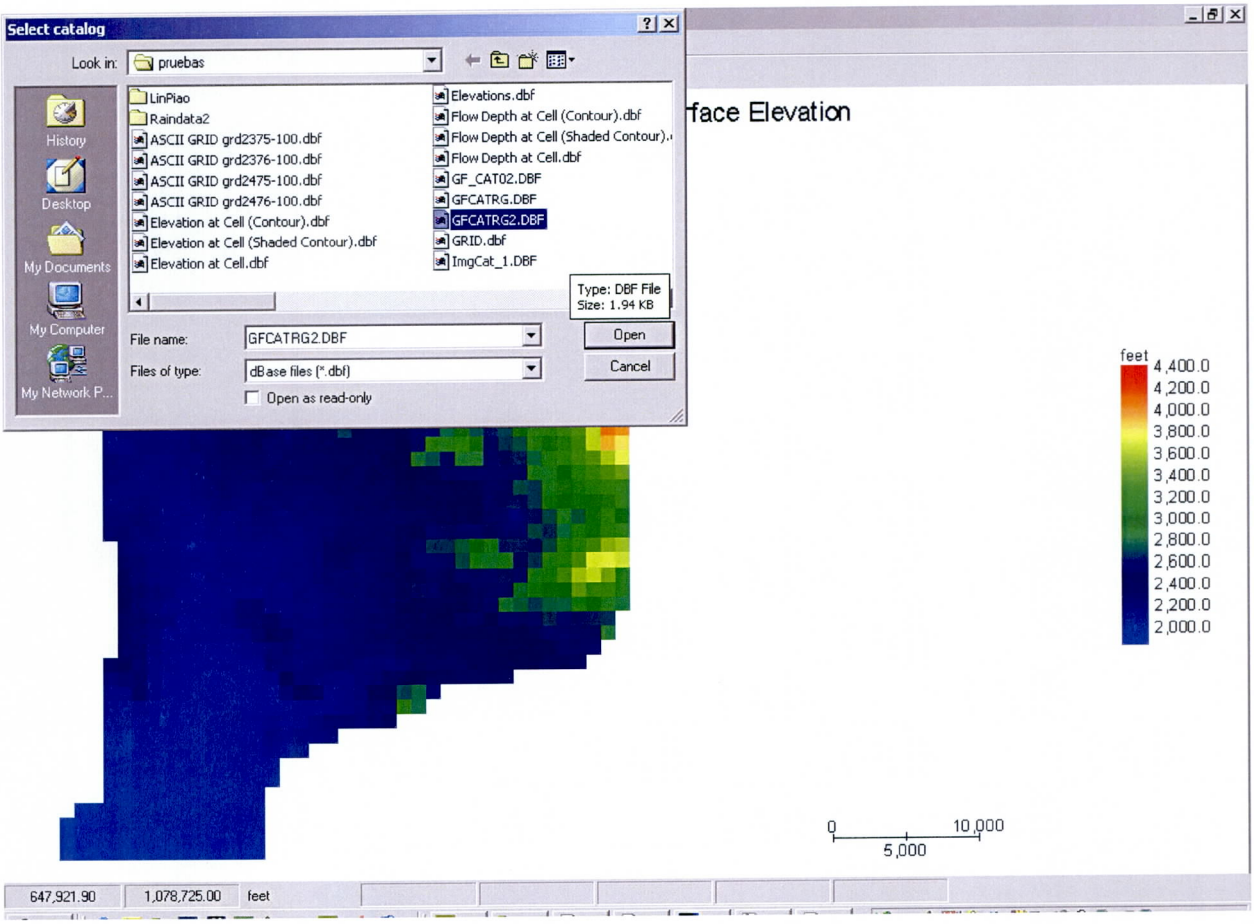


Use this command to import an individual ArcInfo ASCII grid files. Several grid files can be imported. The new points are added to the existing data. This data can also mixed or combined with the DTM points.

ArcInfo ASCII Grid File Catalogue (File Menu)



This command will import several ArcInfo ASCII grid files stored in any subdirectory. First draw a polygon on the working region and then select a catalog file.

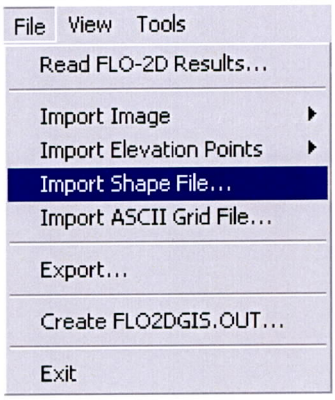


The catalog file may be in DBASE or ASCII format and has the following format:

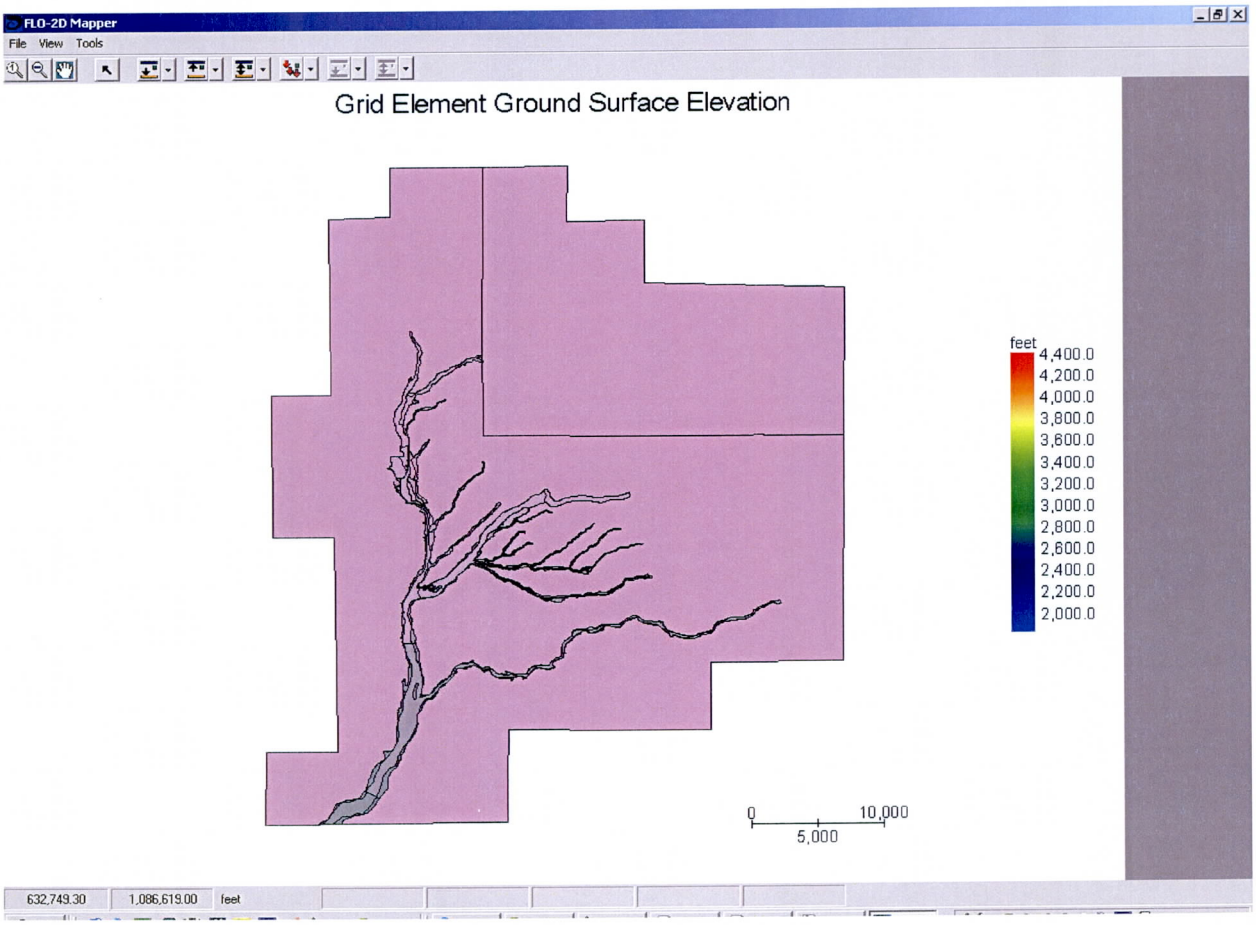
ASCII_GF	XMIN	YMIN	XMAX	YMAX
C:\Projects\MaricopaCounty\Data\grd2375-100.asc	640000	1000000	670000	1400000
C:\ Projects \MaricopaCounty\Data\grd2376-100.asc	500000	1000000	900000	1500000
\\Agua\IMF\Publico IMF\grd2476-100.asc	510000	1000000	740000	1510000

The first column is the file name including its path. The following four columns are the data limits in each file. *Mapper* will find all data files that are contained or intersect the user defined polygon and retrieve the relevant files.

Import Shape File Command (File Menu)



This command allows importing ESRI shape files. The shape file data map can be related to soil properties, land use, vegetation, Manning roughness coefficients or similar data. Once read, the shape file is displayed in the screen like the following example from Maricopa County Manning coefficients shape file.

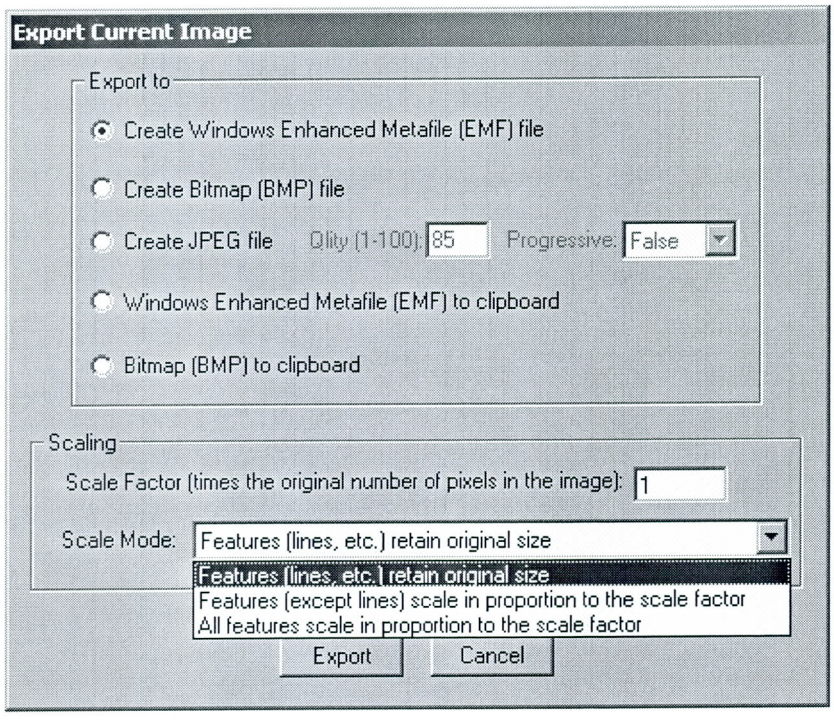


Import ASCII Grid File Command (File Menu)

This command will import ArcInfo ASCII grid files stored in any subdirectory.

Export...Command (File Menu)

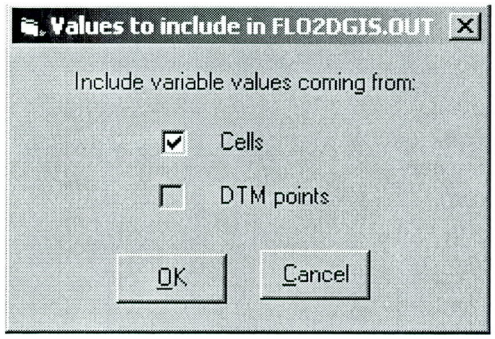
Using this command you can export the current screen view to different image format. Click on the Export Command the following dialogue box will appear:



In this dialog box, image export formats can be selected such as Bitmap, JPEG, Windows Matafile, etc. The scale factor of the image can be adjusted for better printing quality. When you click on the Export button, input the image file name and select the directory to save the image in the subdirectory/file dialogue box.

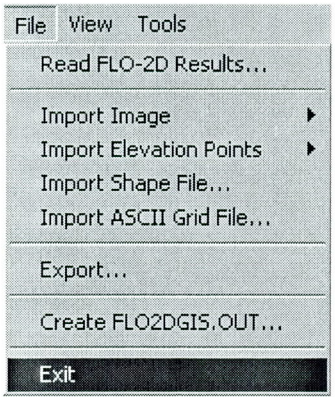
Create FLO2DGIS.OUT... Command (File Menu)

Use this command to create a GIS output file. The FLO2DGIS.OUT file contains the x- and y-coordinates of each point and their associated ground elevation, water surface elevation, flow depth and water velocity. It is necessary to first interpolate the flow depth from the grid element water surface elevation and DTM ground points using the Interpolate to DTM Points command in the Tools Menu. This GIS output file can be imported to ArcView™. Mapper presents a dialog box to select the values to be written into the file:



Values assigned to the DTM points can only be written to the FLO2DGIS.OUT file if the flow depth interpolation from the grid element water surface and DTM points has been performed.

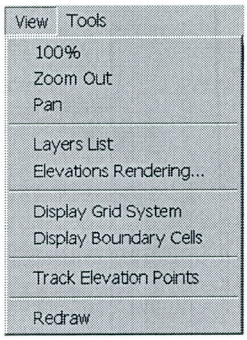
Exit (File Menu)



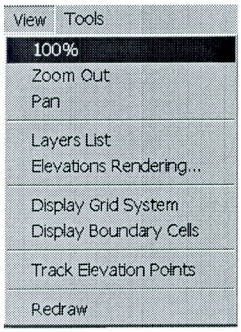
This command ends the MAPPER session. You can also use the *Close* command on the application Control menu.

Shortcuts: Mouse: double-click the application Control-menu button.
Keys: Alt-F4

View Menu

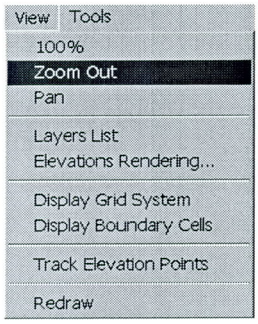


100% Command (View Menu)



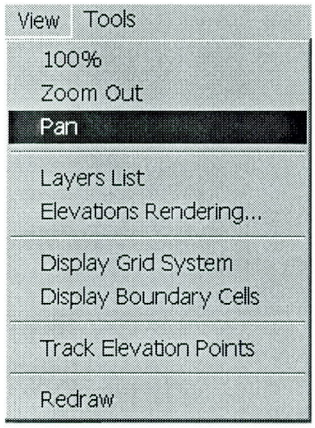
Use the *100%* command to revert the working region to its original size. You can also use the *100%* toolbar icon for the same purpose. To use the *100%* command (or toolbar icon), click on the icon. To Zoom-in (magnification of the working region) just click in the working region and drag the mouse to the for magnification desired.

Zoom Out Command (View Menu)



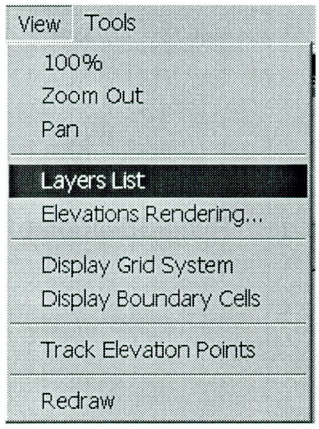
Use the *Zoom Out* command to return the working region to its previous zoom extent.

Pan Command (View Menu)



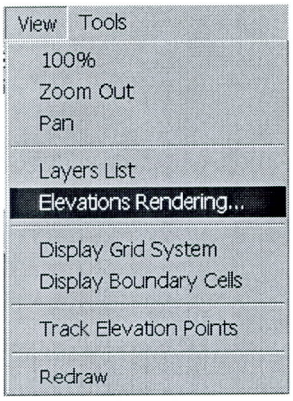
Use the *Pan* command to move and display the working region view. When activated the mouse cursor changes to *Pan* mode and you can drag and click to pan around. The *Pan* toolbar icon achieves the same result.

Layers List Command (View Menu)

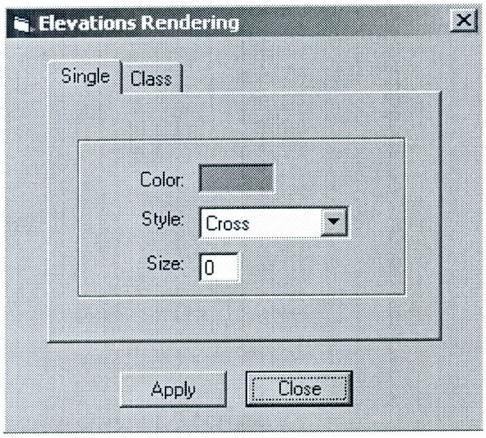


This command is used to manage existing layers. Please refer to GDS Layer List Command for instructions on its application. The commands are identical.

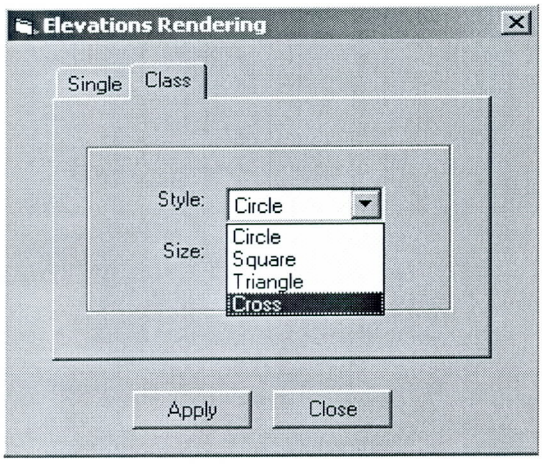
Elevation Rendering... Command (View Menu)



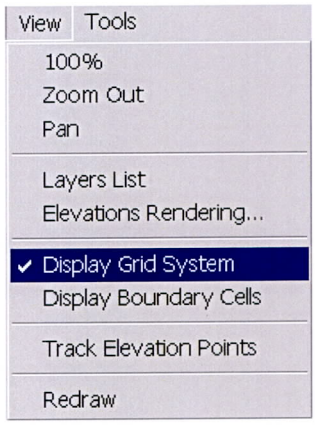
This command may be used to select how the elevation points are displayed.



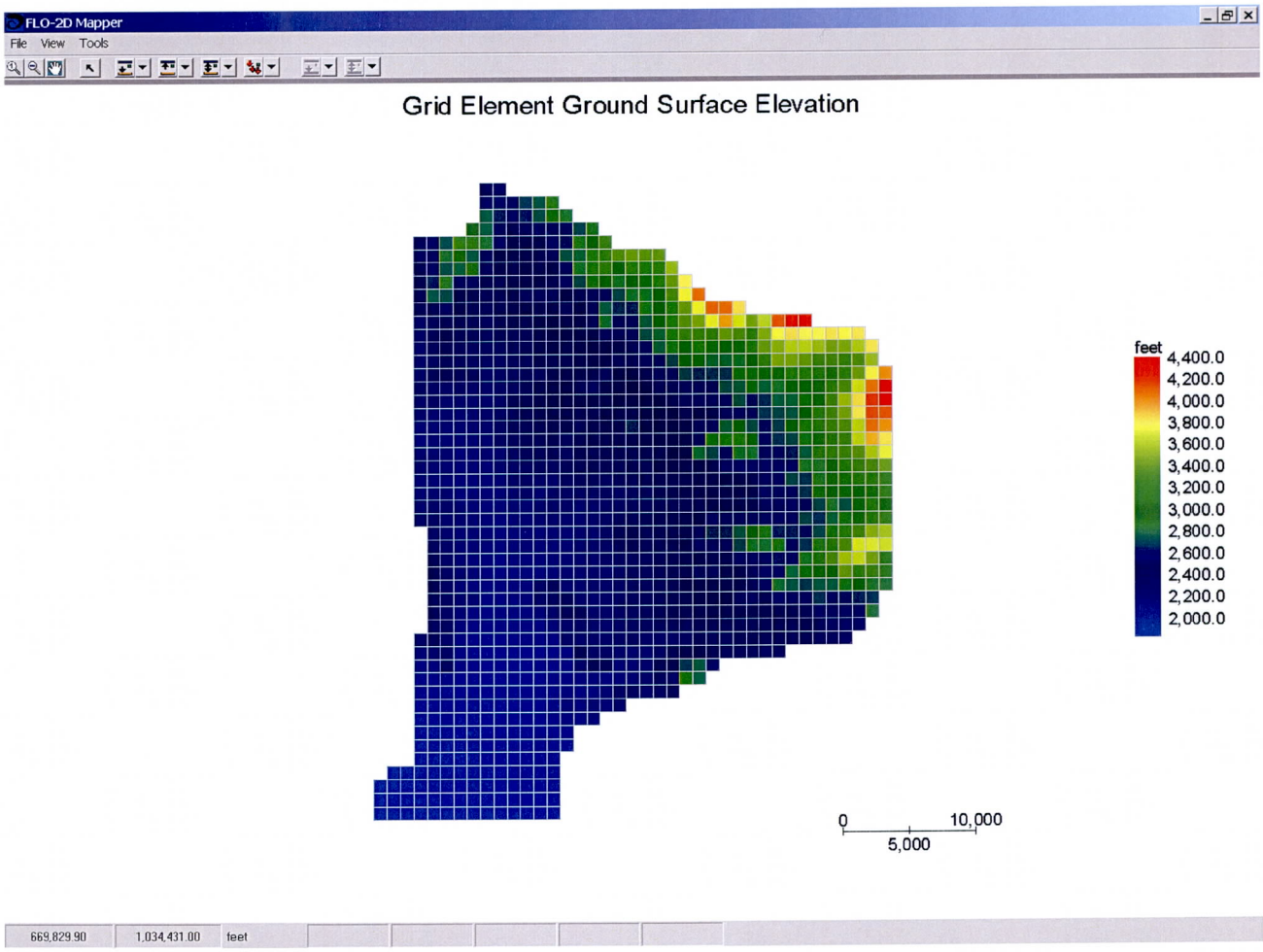
The "Single" tab selects a single color, style, and size for all elevations. The "Class" tab selects a style (point descriptor) and size for the points. A palette of 256 colors are available to be assigned to the point elevation.



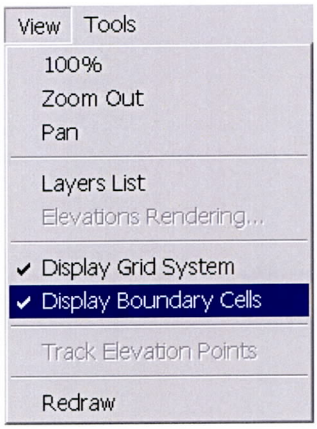
Display Grid System Command (View Menu)



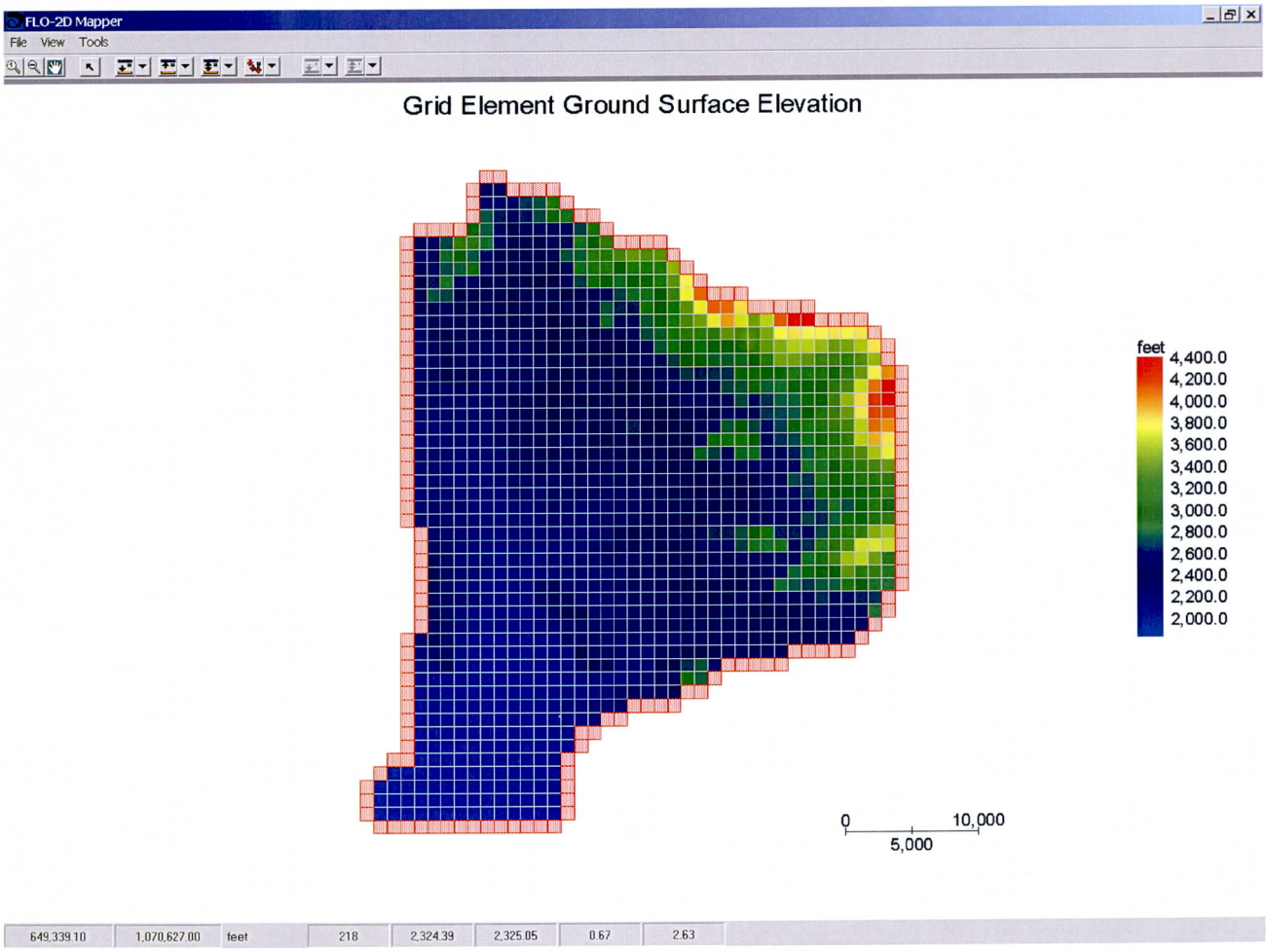
To view the *FLO-2D* grid system as an overlaid image, click on this command.



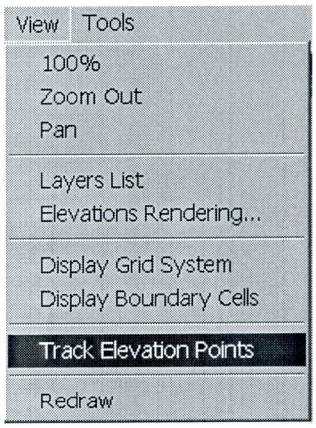
Display Boundary Cells Command (View Menu)



This command displays the *FLO-2D* grid boundary cells.



Track Elevation Points Command (View Menu)

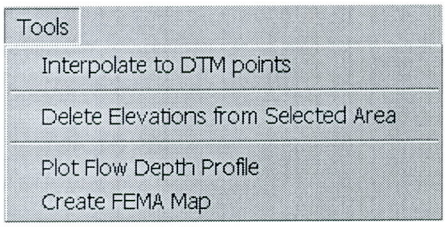


With this command, the elevation of a particular DTM point is presented in a dialog box. When the mouse pointer is placed on the vicinity of a point, its elevation is displayed in a text box in the top right corner of the main window:

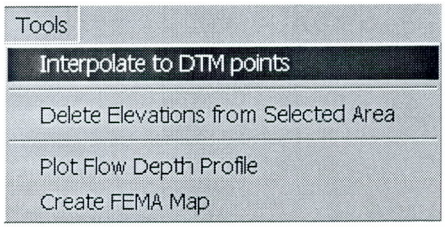
Redraw Command (View Menu)

Use this command to redraw the visual objects in the *working region.*

Tools Menu

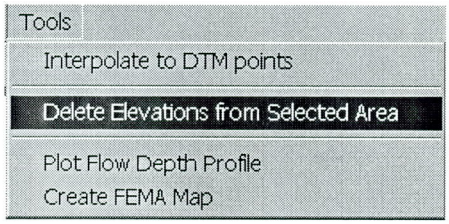


Interpolate to DTM Points Command (Tools Menu)



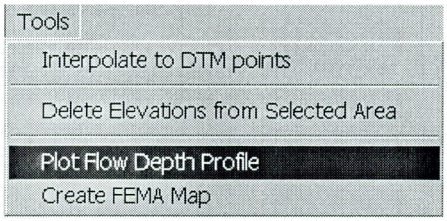
Use this command to interpolate the DTM points flow depth. The flow depth is computed by subtracting the DTM ground surface elevation from the grid element water surface elevation for each DTM point within the grid element. This creates a data base of flows depths associated with every DTM point. It is necessary to first import the DTM points using the Import DTM Elevation Points command in the File Menu.

Delete Elevations From Selected Area Command (Tools Menu)

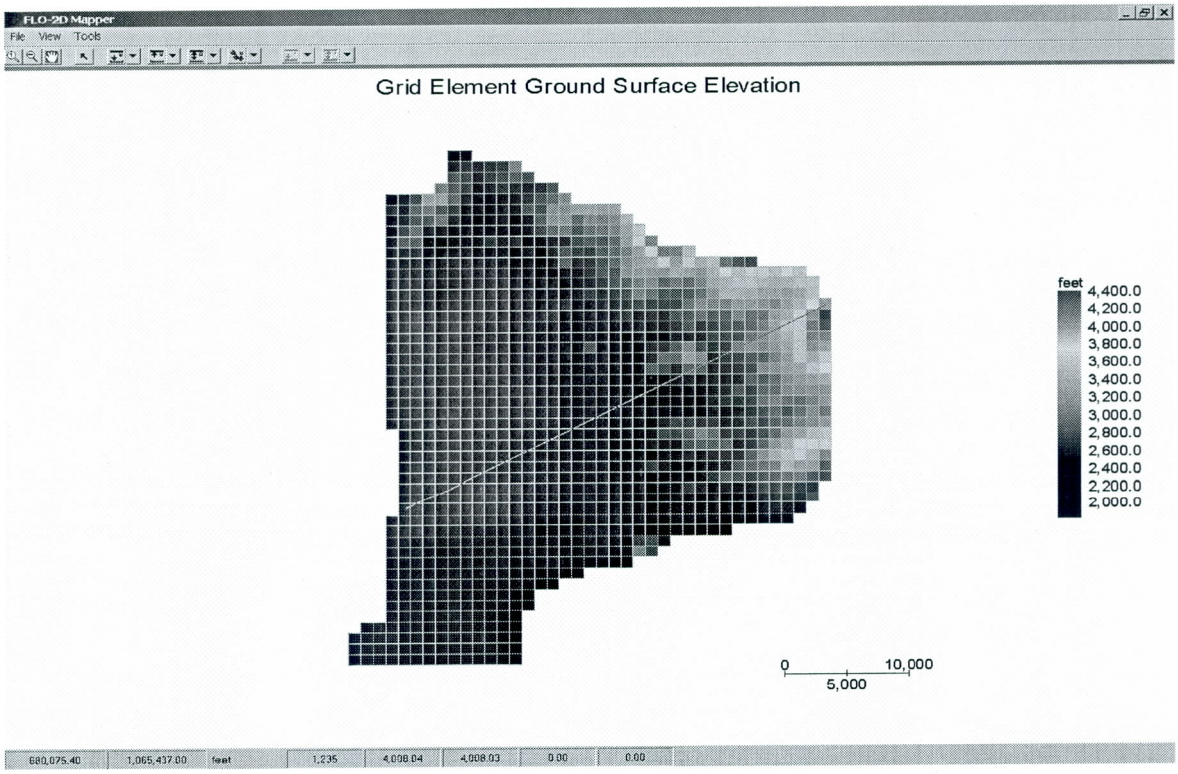


This command will delete unwanted elevation points from the screen. It is identical to the GDS command. Please refer to the GDS section for the command instructions.

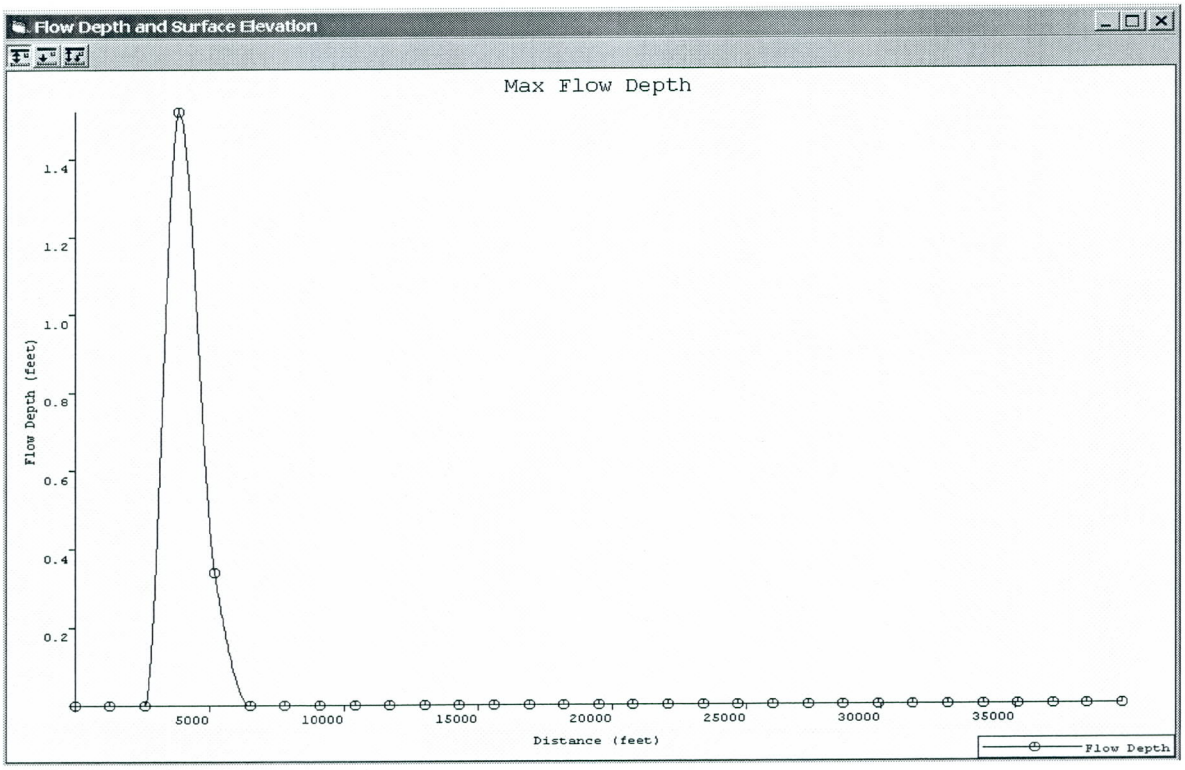
Plot Flow Depth Profile Command (Tools Menu)



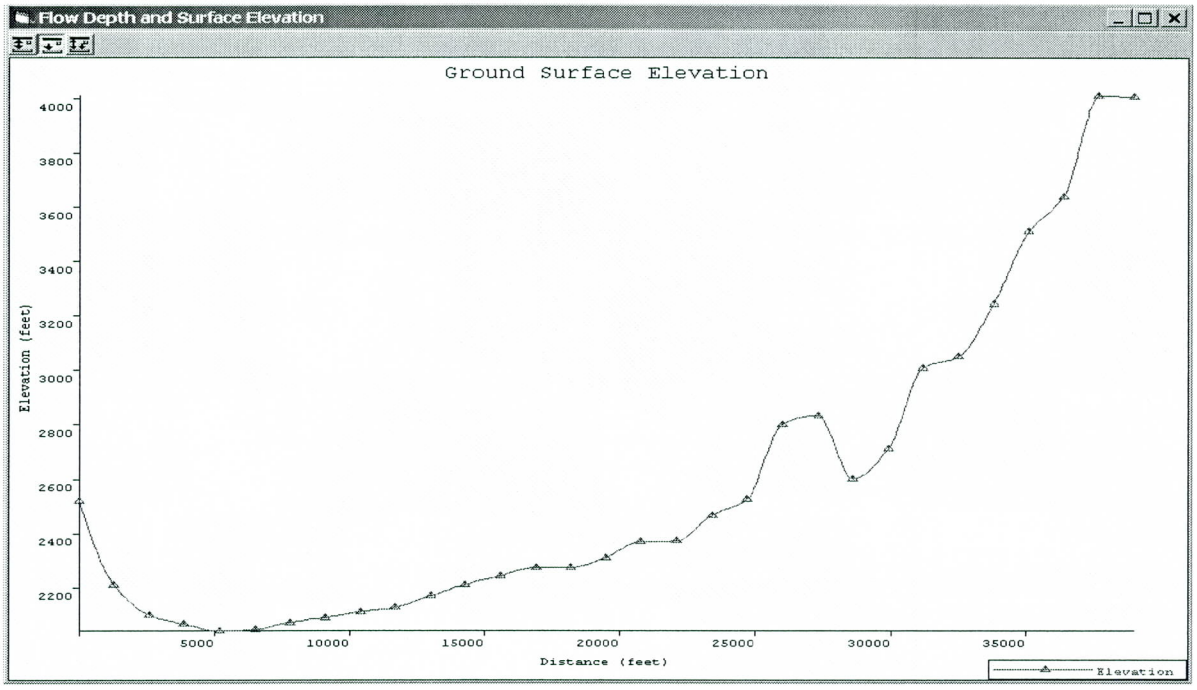
This feature will plot profiles of maximum depths maximum elevations and bed elevation profiles. Click on *Plot Flow Depth Profile* command and you draw a line where you want to create the profile. To draw the profile line, click on the first point and locate the mouse cursor on the second point and double click. The profile line must be contained in the computational domain.



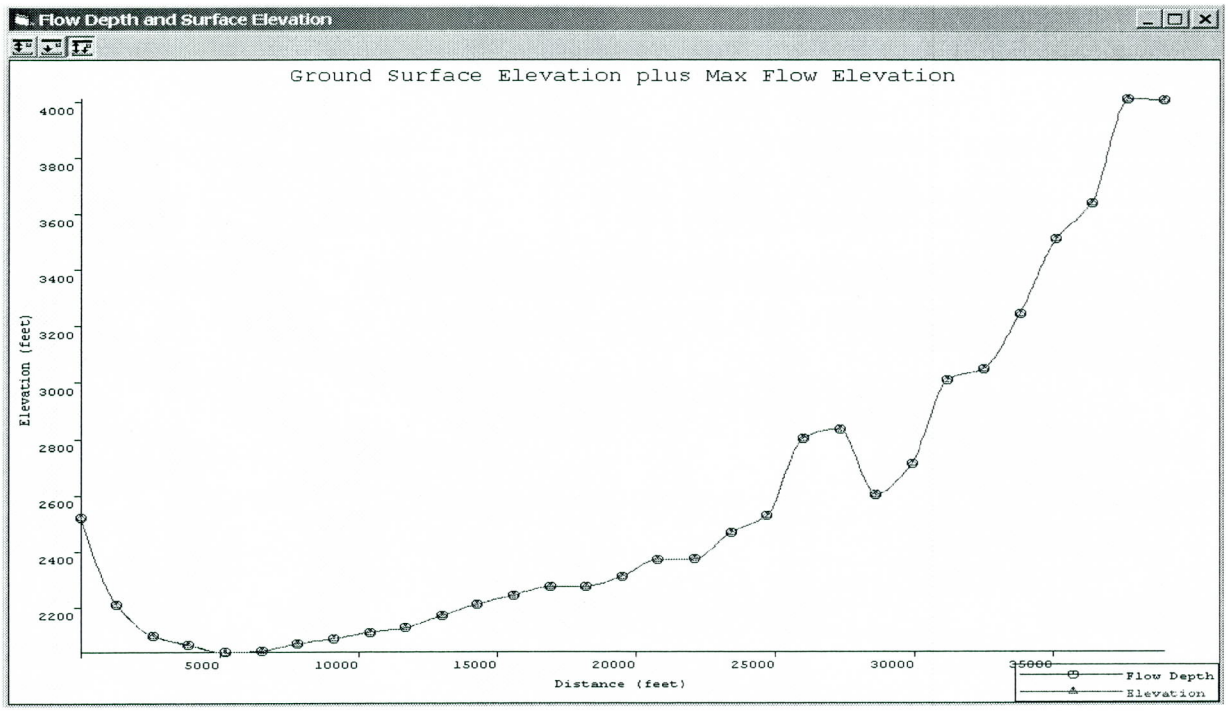
After double clicking on the second point of the line, the following window appears showing the maximum depth profile:



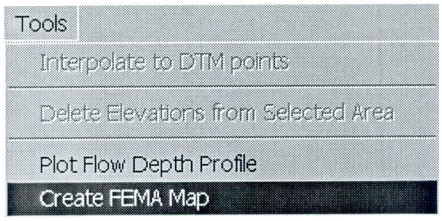
Note that in the upper left corner of the window there are three icons: . The first icon the system plots Maximum Flow Depth profile (shown above). The second icon will plot the Ground Surface Elevation profile:



And the third icon is used to plot the combined maximum depth and ground elevation profile:



Create FEMA map Command (Tools Menu)



This command automates the process to create Federal Emergency Management Agency (FEMA) FIRM maps (Flood Insurance Rate Maps). Select the *Create FEMA map* option from the *Tools* menu and the system rescales the maximum depth plot and draws a template frame. The template includes a left column for text and a right column with two parts: The upper part for the legend/text and the bottom part for the FEMA logo and map ID. The user can edit, move, cut and paste text to both columns. If text already exists, clicking on the text region will change its position. By clicking on the text region with the right mouse button, a pop-up menu will appear with the options: *Delete text*, and *Edit text*. The *Edit text* option leads to a dialog box that enables new text entry or copy and paste existing text. If there is no text in the text region, the user may enter text. In the lower right region, the user can insert an image that represents the FEMA office or other organization logo. After map template format and text has been completed, it can be exported to one of the supported images for final printing.

Toolbar Plotting Buttons

The toolbar contains the buttons to generate plots or contour maps of the following variables:

Grid Element Ground Surface Elevation

Grid Element Maximum Water Surface Elevation

Grid Element Maximum Flow Depth

Grid Element Maximum Velocity

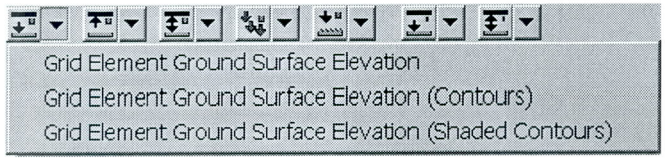
Grid Element Maximum Deposition

DTM Point Ground Surface Elevation

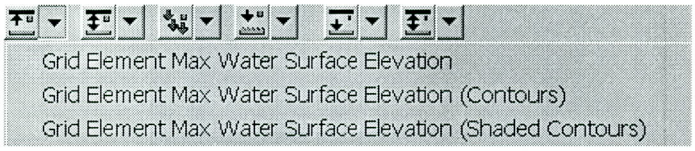
DTM Point Maximum Flow Depth

Every icon has a drop down button that will display a menu for each available plot as follows:

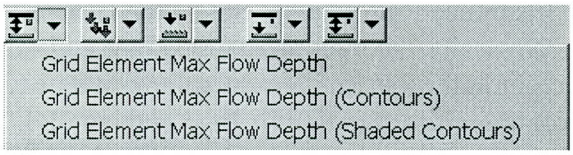
Plot of the grid element ground surface elevation



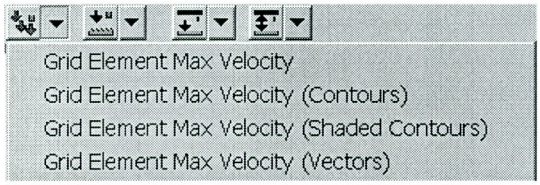
Plot of the grid element maximum water surface elevation



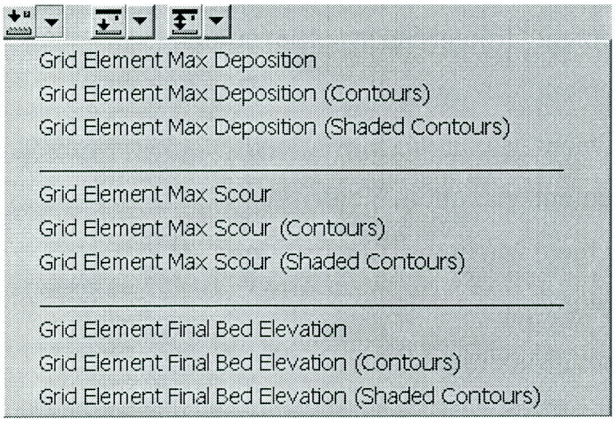
Plot of the grid element maximum flow depth



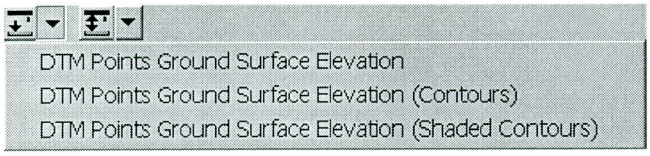
Plot of the grid element maximum velocity



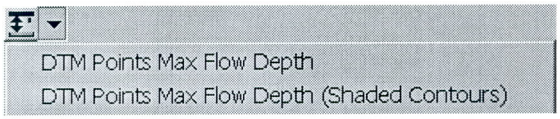
Plot of the grid element maximum deposition, maximum scour and final bed elevation



Plot of the DTM point ground surface elevation



Plots of the DTM point maximum flow depth

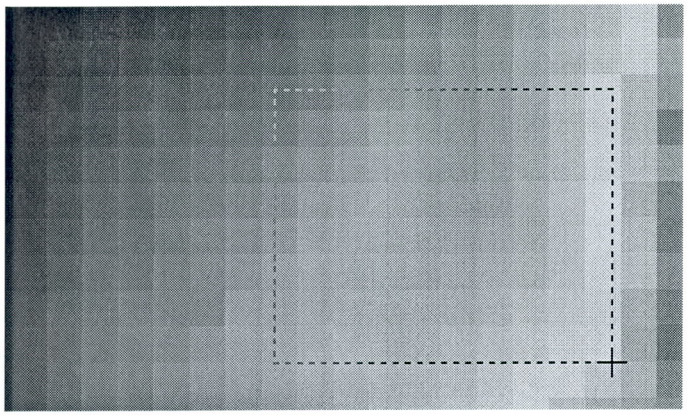


The left side of the bottom tool bar also has data boxes that display the x-coordinate, y-coordinate, grid element number, ground and water surface elevation and maximum flow depth and velocity when the mouse cursor passes over the grid element point on the flow domain.

Mouse Button Commands

Zoom in (left mouse button)

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



Color scale repositioning (left mouse button)

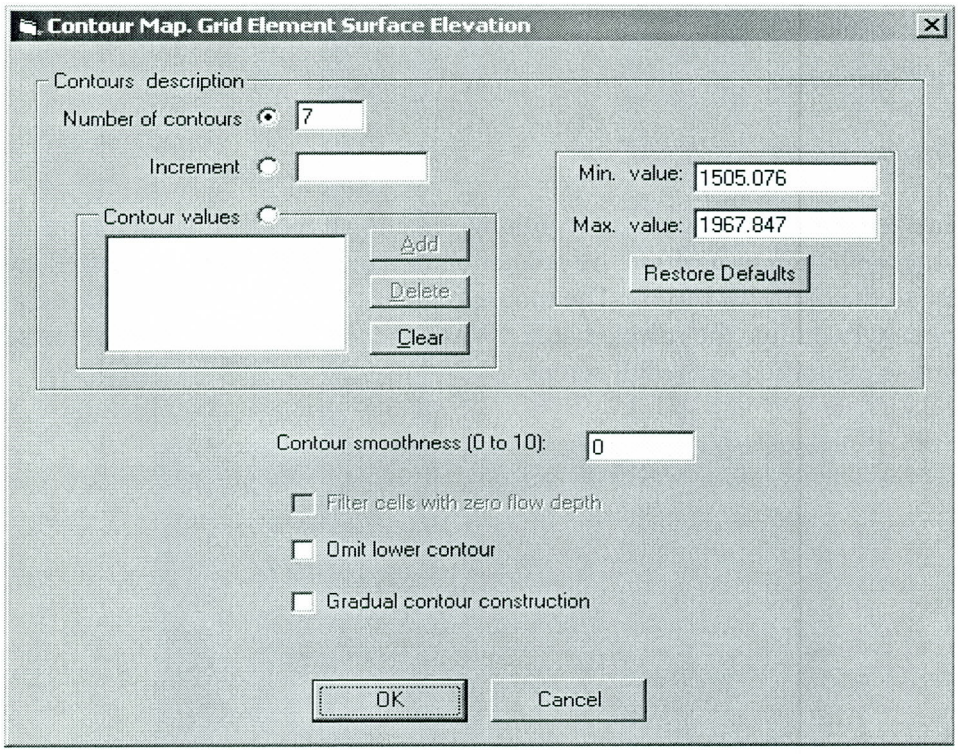
A color scale is usually placed at the right of the graphs. The user can reposition it by left clicking the mouse on the scale and dragging it to the desired position.

Distance scale repositioning (left mouse button)

A distance scale is usually placed at the right of the graph below the color scale. The user can reposition it by left clicking the mouse on the scale and dragging it to the desired position.

Right Click on Contour Graphs

This option is activated by clicking the right mouse button over the displayed plot. Contour intervals and other options can be modified for all contour and shaded contour plots. After clicking the right mouse button on the plot you want to modify, a dialog box appears:



In this dialog box, you can define three options to display contours or shaded contours.

1. Define the *Number of contours* and the contours are displayed for equal increments between the minimum (*Min. value*) and maximum (*Max. value*).

2. Specify a contour *Increment* and the contours are displayed in successive increments starting with the minimum value (*Min. value*).

3. Specify a list of *Contour values* you want to display. You may *Add* or *Delete* any value from the list and *Clear* the whole list.

The option "*Contour smoothness (0 to 10)*" selects a level of resolution for the contours. A higher for the contour smoothness requires more points for the contour interpolation, but it will also significantly increase the contour computation time.

When the option "*Omit lower contour*" is selected, the lower contour is not drawn.

The "*Gradual contour construction*" option selects an alternative contour building algorithm, that will produce better contour graphics for certain plots.

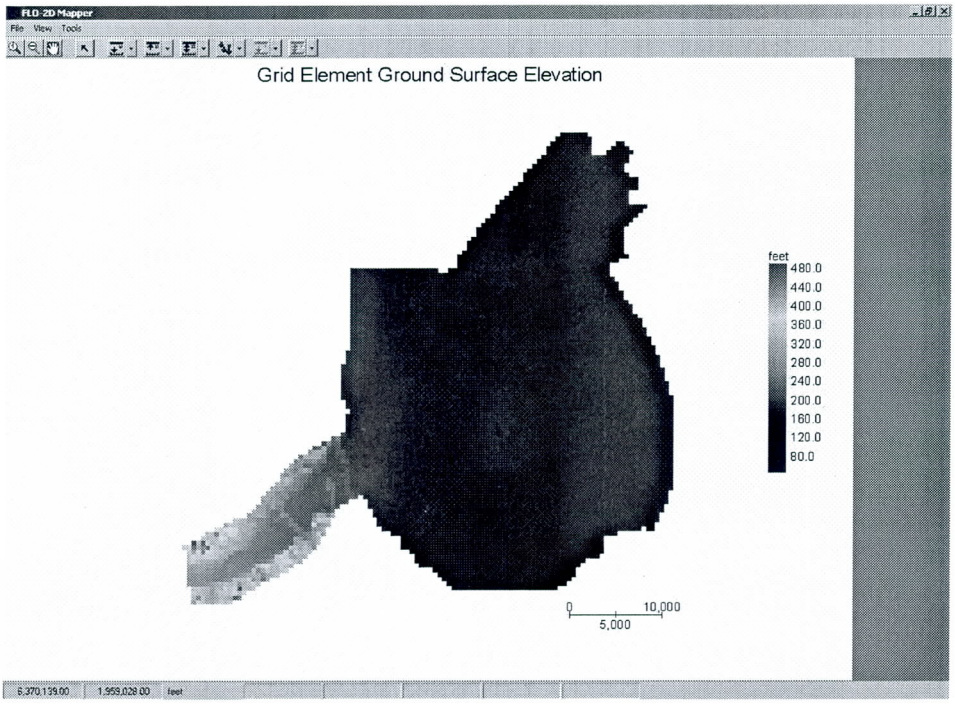
The option "*Filter cells with zero flow depth*" works only for flow depth contour plots. When this option is selected, only the grid elements with flow depth greater than zero are considered when developing the contours.

The plot options are similar for all the hydraulic variables.

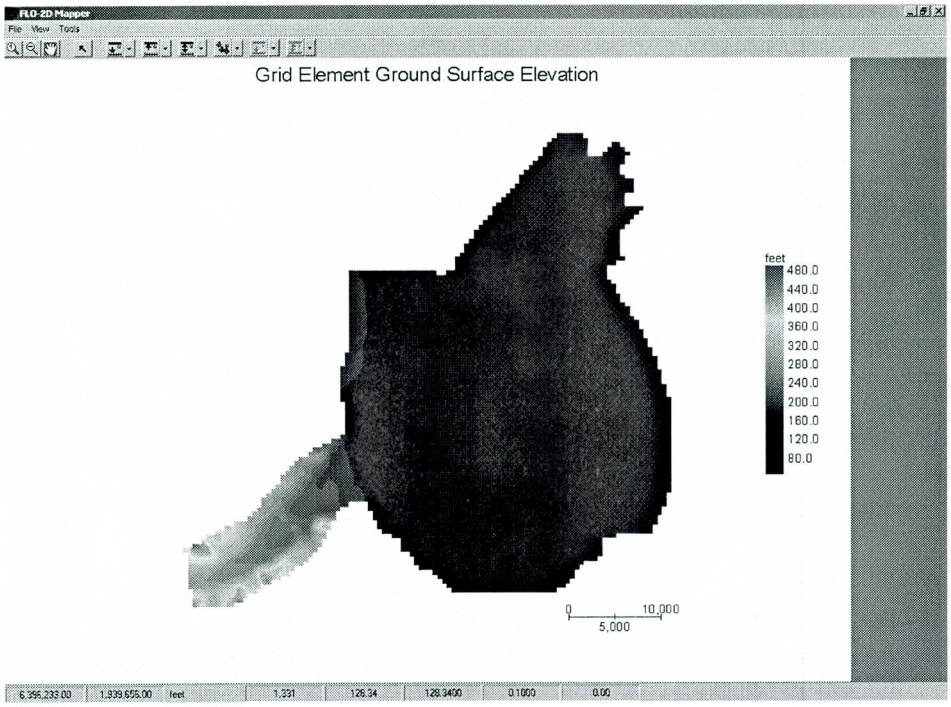
7.2.5 Plot Examples

The following are some examples of the MAPPER graphics.

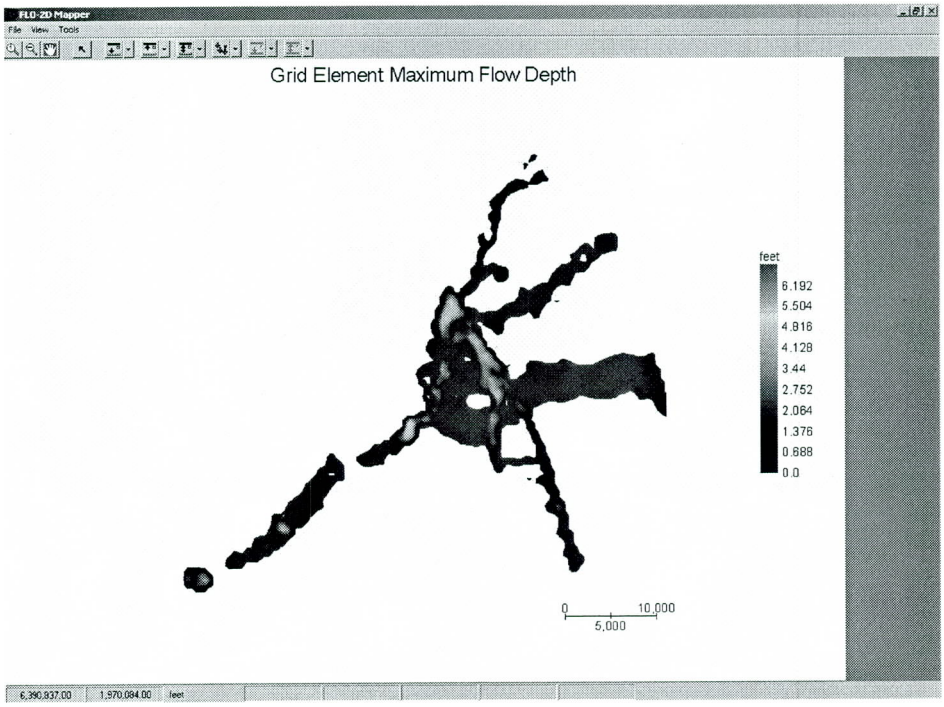
Grid Element Ground Surface Elevation



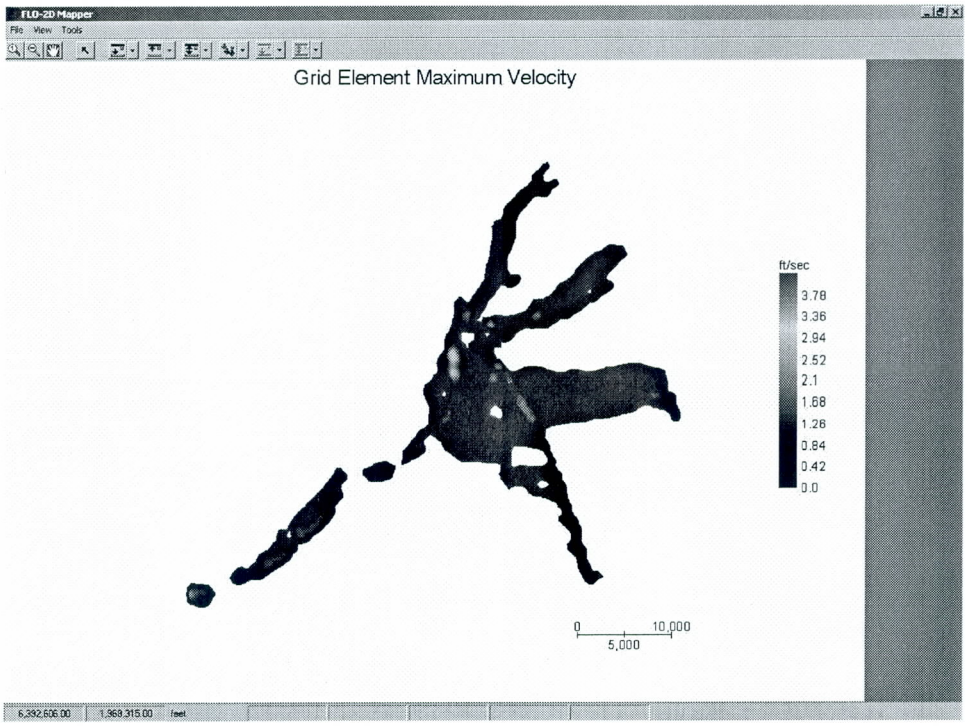
Grid Element Ground Surface Elevation (Shaded Contour Map)



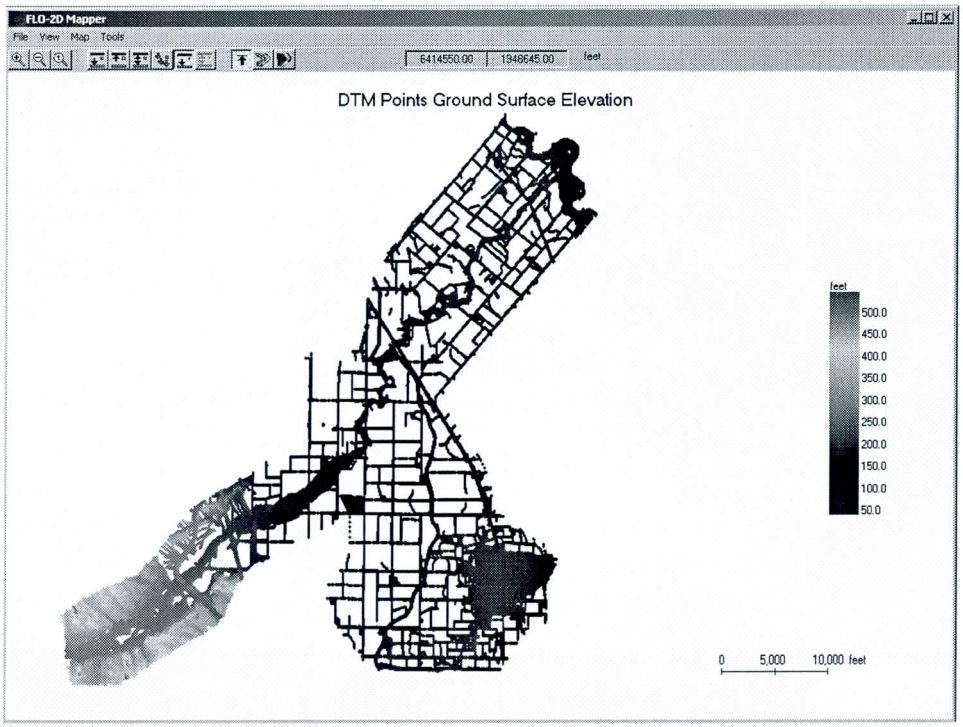
Grid Element Maximum Flow Depth (Shaded Contour Map)



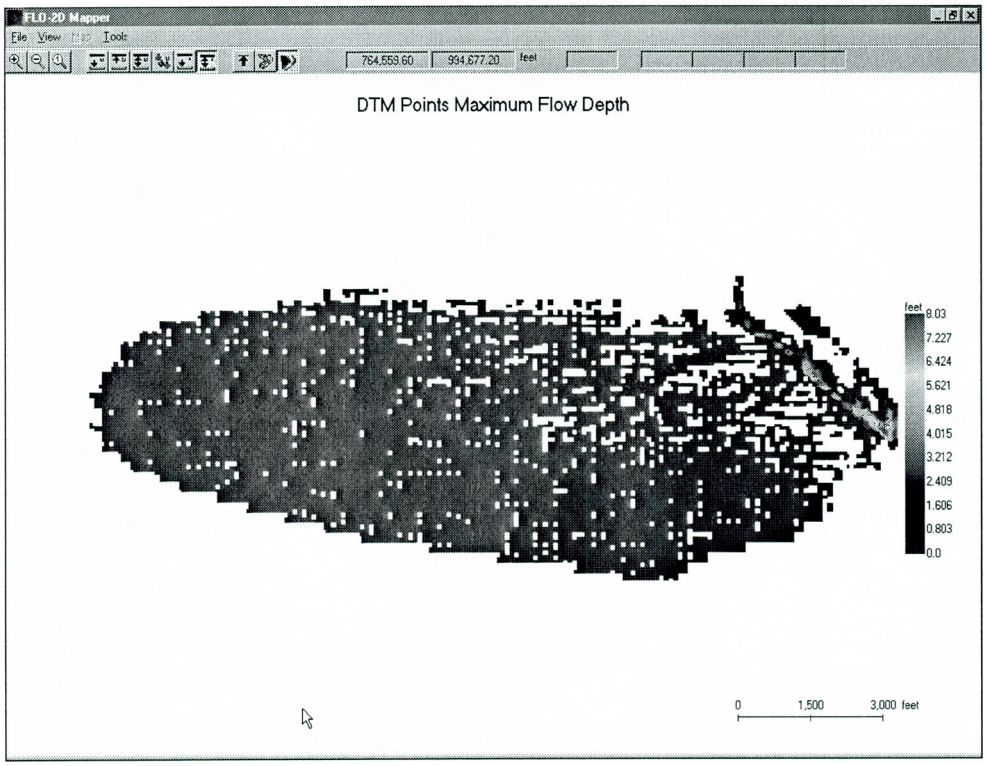
Grid Element Maximum Velocity (Shaded Contour Map)



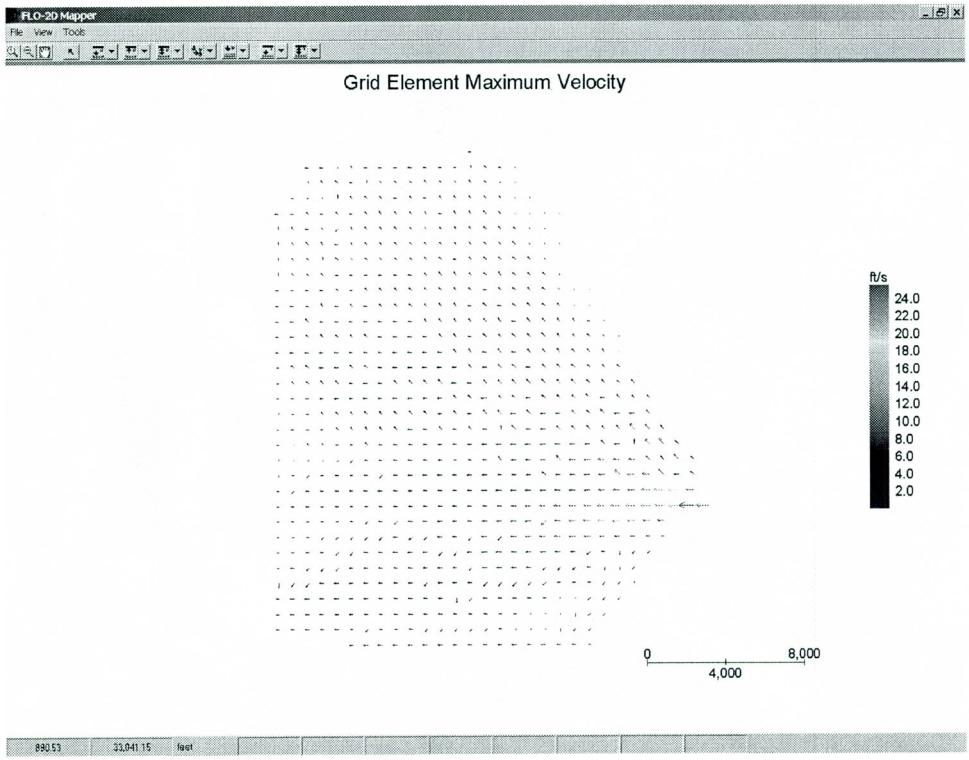
DTM Point Ground Surface Elevation



DTM Point Ground Surface Elevation



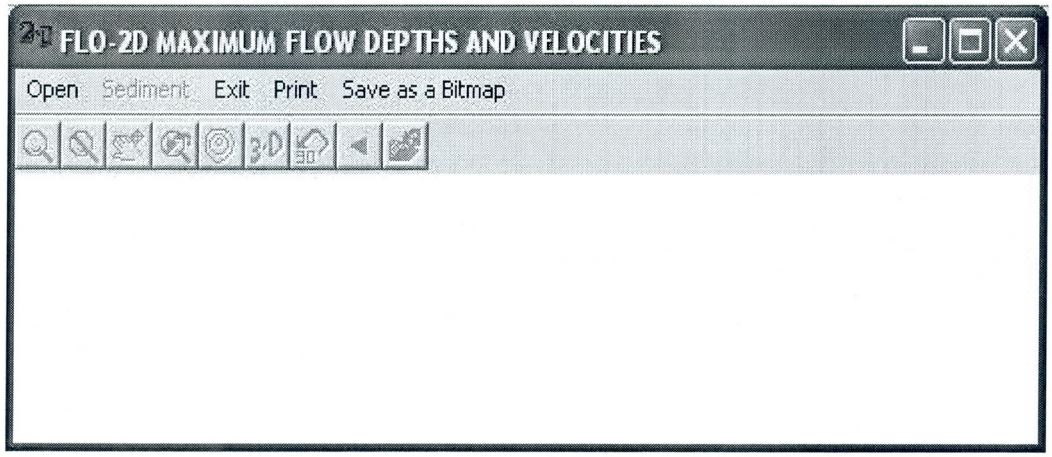
DTM Points Maximum Flow Depth



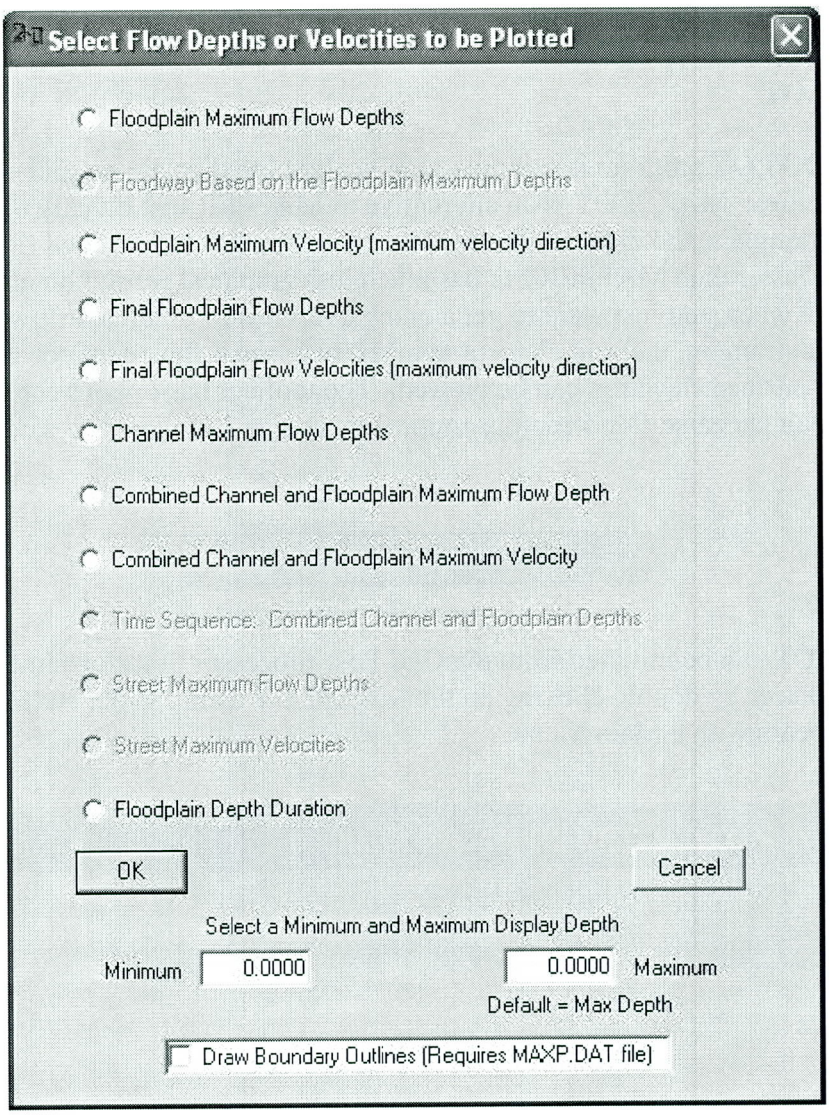
7.3 MAXPLOT

The MAXPLOT program is a simple graphical tool to display the grid element maximum depths and velocities. MAXPLOT is an alternative to MAPPER that quickly displays plots of the maximum floodplain and channel depths, maximum street velocity, final floodplain depths and others. It is faster than MAPPER but has much less graphical resolution and display options. Use MAXPLOT when you just want to get a quick overview of predicted flow depths and velocities. By zooming in, the grid element number, maximum flow depth or velocity and the maximum water surface elevation can be viewed. The tool bar has options for view extents, previous view, pan, a coarse flood contour routine, an coarse 3-D view and save view as a bitmap.

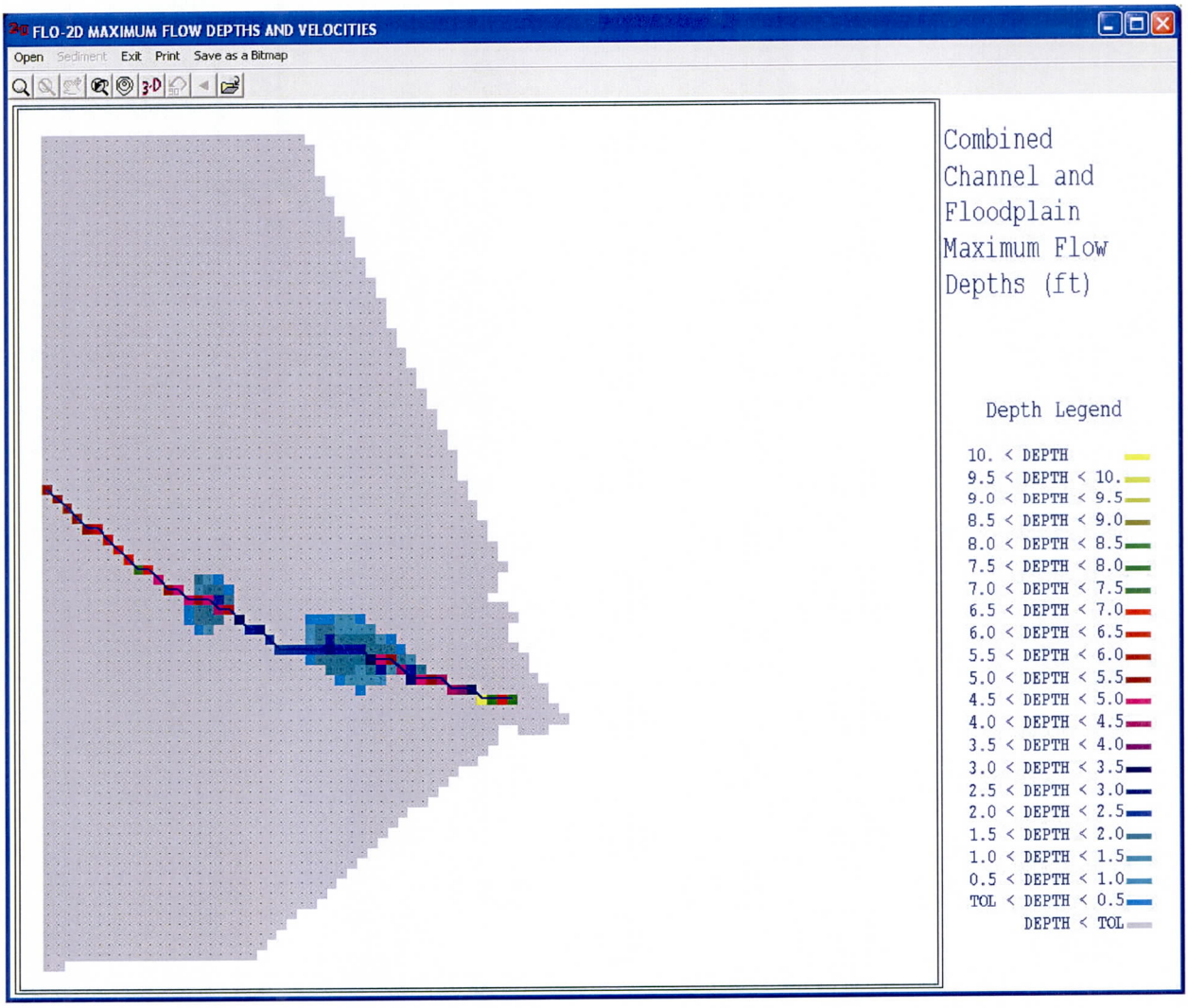
MAXPLOT can be initiated from the GUI Post-processor pull-down menu or it can be started from Explorer by double clicking on the MAXPLOT name. After starting MAXPLOT a blank screen appears with a Main Menu:



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 'OK'. You can also set limits on the minimum and maximum depths or velocities to display. The following plot displays the combined channel and floodplain maximum flow depth for the Monroe project example.



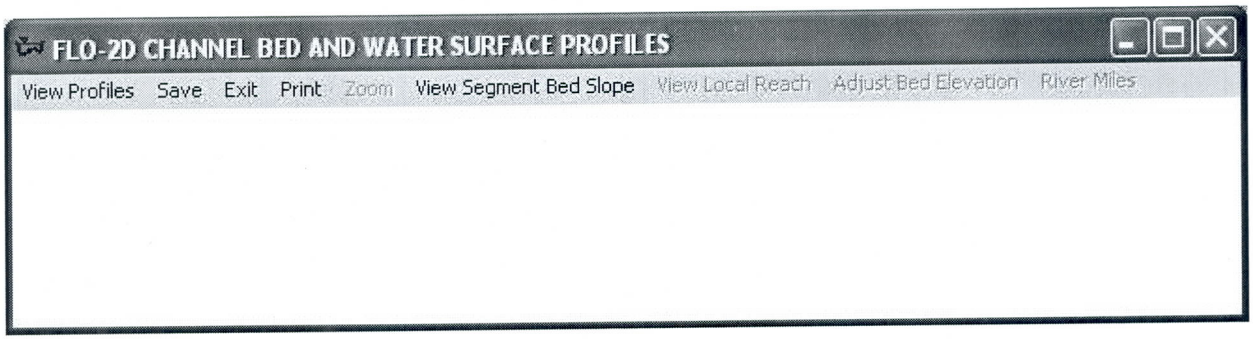
MAXPLOT also has an option to plot a floodplain boundary such as the area of inundation, level location, or other linear boundary (see the bottom of the above dialog box). The MAXP.DAT file contains the boundary line locus of points must be prepared to display the boundary. A series of line boundaries can be prepared. Each set of boundary points will constitute a separate line. The optional MAXP.DAT file format is:

Line 1. Number of individual boundary units. Each boundary unit will constitute 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.

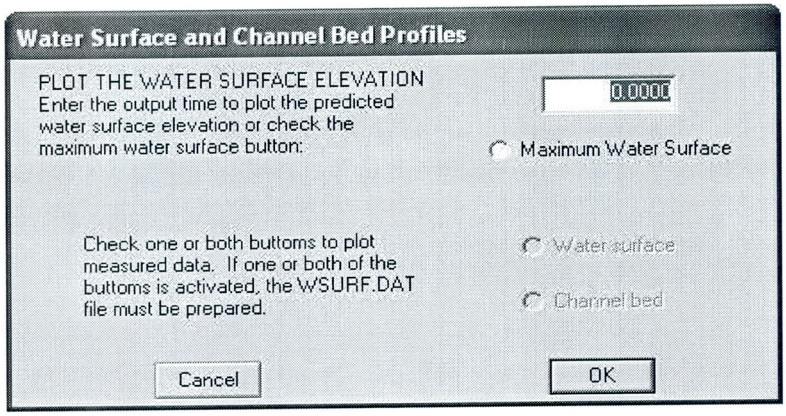
Repeat Lines 2 and 3 for each boundary line set of points.

7.4 PROFILES

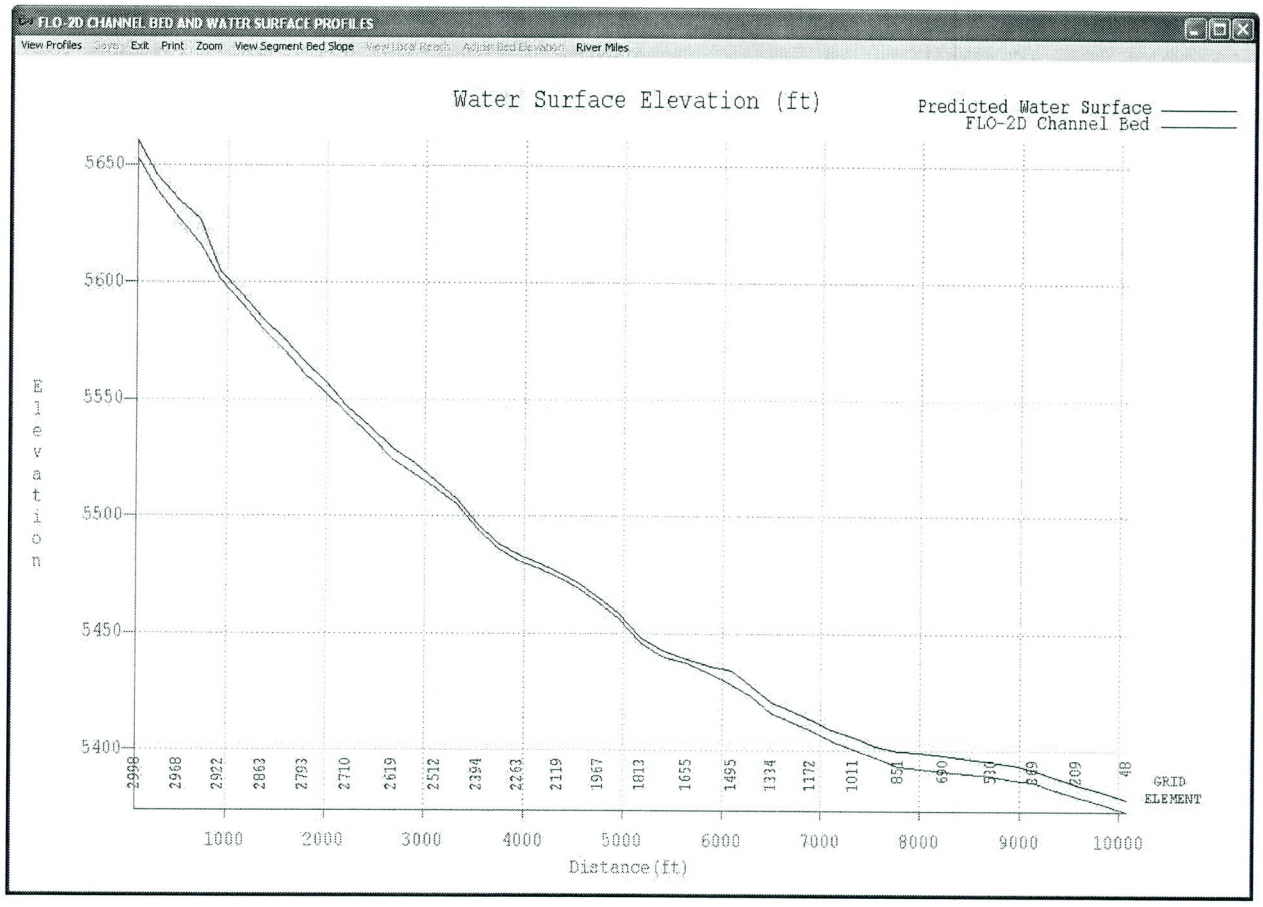
Profiles serves the dual purpose of being both a pre- and post-processor program. As a post-processor program, it will display a channel water surface and bed elevation for any FLO-2D simulation output interval. In order to run the PROFILES program and view the predicted water surface elevation, it is necessary to run a FLO-2D channel simulation first. The PROFILES program has zoom and print options to assist in reviewing the results.



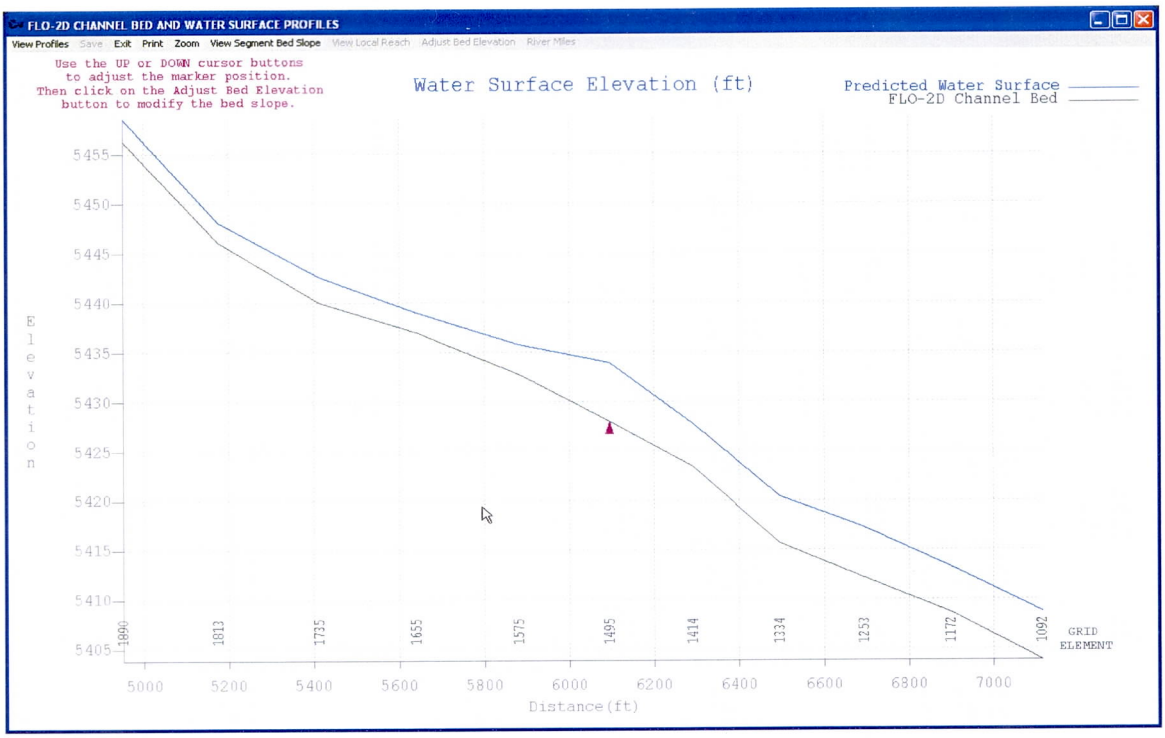
To view the predicted water surface profiles, click on *View Profiles* in the Main Menu and a dialog box appears:



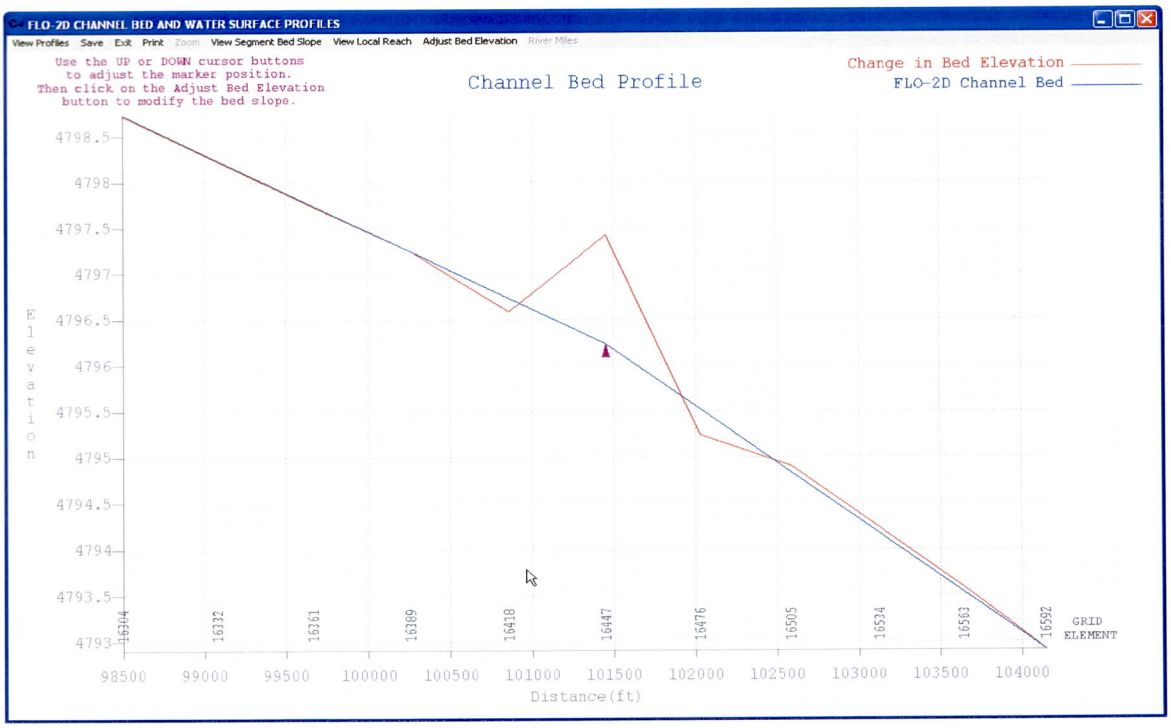
To view the predicted maximum water surface elevation profile, click on the radio button labeled 'Maximum Water Surface' and click 'OK'. You can also plot the water surface at any output interval by entering the time in the text box in the upper right corner. To plot the surveyed or measured, the WSURF.DAT must be prepared. The file format is presented below.



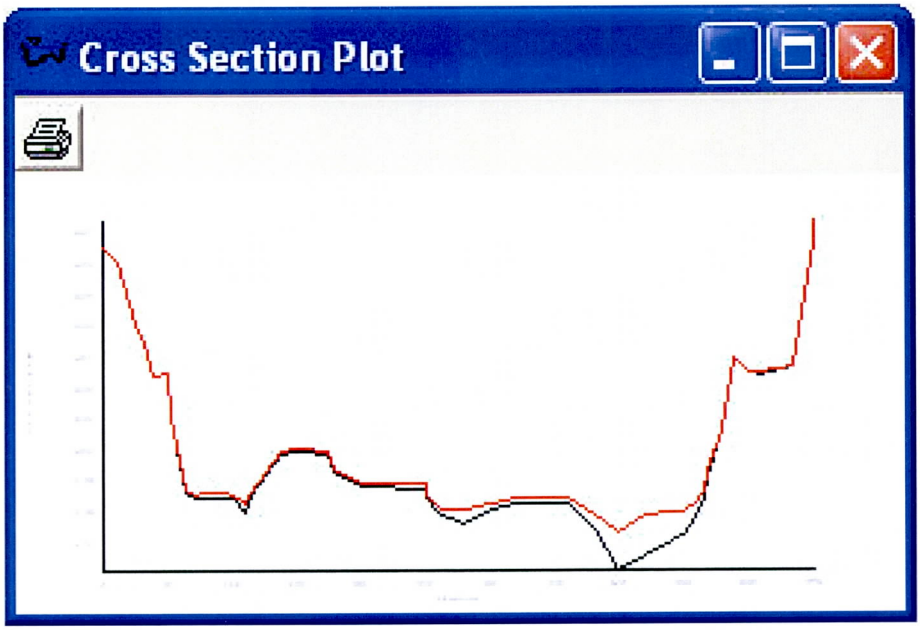
There are several options on the Main Menu. You can zoom in on given river reach, print the image or label the distance along the channel in river miles. The zoom view is shown in the following figure:



If sediment transport has been simulated, PROFILES will plot the final bed elevation and the cross section geometry changes associated with either scour or deposition. The image below displays sediment deposition of approximately 1 ft upstream of a bridge in the Rio Grande.



Non-uniform sediment distribution on the channel bed can be view when the channel flow is simulated with cross section data. The cross section plot below displays the final cross section elevations in red compared to the original elevations in black based on the sediment transport component in the FLO-2D model. This image can be expanded to full view.



The user has an option in the water surface dialog box of plotting surveyed water surface and bed elevations along with the predicted values. To plot the surveyed water surface or channel bed elevation, the WSURF.DAT file must be created in the following format:

Optional WSURF.DAT file format:

Line 1. Number of grid elements with a surveyed water surface or bed elevation.
Line 2 to end. Grid element, measured water surface, surveyed bed elevation in columns.

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.

VIII. CREATING FLOW DEPTH CONTOURS WITH CADD PROGRAMS

The DEPTH.DAT and VELOC.DAT files have been formatted for importation into a CADD drawing to plot the maximum depth or velocity contours on the digitized base map in the CADD drawing. It is suggested that these files be renamed for the various flood simulations and saved before running FLO-2D again. To import these files to a CADD program the following general steps are necessary:

Import Points

There will be some unique 'import points' commands for each CADD program, but these steps are required for most programs.

1. Make a new current layer for the depth or velocity points to be imported.
2. Turn off (freeze) any unnecessary layers.
3. From the appropriate menu, select the import/export option.
4. Select the required data import format (point number, easting, northing, elevation).
5. Entering the subdirectory and file name in import dialog box.
6. When the points are imported, you may have to choose to either append or overwrite the project points. Generally it is more convenient to overwrite the project points and have the grid element numbers start with 1.

The points should now appear in the drawing. If the points do not appear on the screen, check to make sure that the appropriate point menu box is turned 'ON' in the various 'POINTS' options. The elevation precision in two decimal places may also have to be selected.

Create a triangular irregular network (TIN) or surface for the flow depth or velocity

1. From the TIN or SURF from the appropriate (DTM) menu.
2. Create a new surface name.
3. Select the 'POINTS' option for importing the data to the TIN or surface option.
4. Chose an option for importing the point data. For example, import the point data by range or by window in the drawing.
5. Create the tin or surface.

Create Contours

1. Select contours option from the appropriate DTM menu.
2. Select the surface created by the TIN or a new surface.
3. When the contour menu box appears, select a contour interval and layer name for the interval contours and highlighted contours. At this point, you may have to select other contour options such as curve smoothing and starting/ending elevations.
4. After the contours are created, you may want to adjust the contour layer color.
5. Label contours with elevations as needed.
6. Create breaklines or other attributes to define channels or other model components.

The contour spacing is interpolated based on the grid spacing or the DTM point spacing. If detailed flood hazard delineation maps are being created, the maximum flow depth contours plotted in the CADD program should be reviewed carefully. Contours lines may be drawn through buildings, across levees and berms, or through other features than will not be inundated. The contours lines should be adjusted in the drawing to account for the failure of the CADD contour routine to recognize these attributes.

The SEDFP.OUT and SEDCHAN.OUT files showing the final bed elevation changes for the floodplain and channel can also be plotted. Each of these files has an additional two columns, which represent the maximum and minimum bed changes (deposition and scour). The file format is:

> grid element, x-coordinate, y-coordinate, maximum elevation change (deposition in ft), minimum elevation change (scour in ft), final bed elevation change (ft).

To plot the final elevation change remove the two columns with the maximum elevation change and minimum elevation change data. This can be accomplished in a text editor.

IX. DEBUGGING AND TROUBLESHOOTING THE DATA FILES

The instructional comments at the end of each file description contain a number of guidelines to assist the user in creating the data files. These comments should be carefully reviewed when preparing each file. Start simple and build on the flood model. After all the data files have been prepared, setup an initial run for a basic overland flood simulation without any other components. Review the results and if you have problems consult the following information.

The first step in debugging a new set of data files is to turn on the IBACKUP variable in the CONT.DAT file to create a set of *.BAC data files. These files are replicates of your data files and will indicate if the computer is reading the data files correctly. The backup file should be identical to the original data file except for spacing. If the program terminates before reaching the first output interval timestep, there is probably an error in the data files. Start by checking the *.BAC files one by one. If one of the files is not complete, this may be the location of the data error.

The most common error is missing data that will invoke an end-of-file error statement to the screen. End-of-file messages are gradually being developed into specific error messages that are report to the ERROR.CHK. This error occurs when the user indicates a prescribed number of data pairs, but then leaves out one or more of the data entries. Most of the counting variables have been eliminated in the model, but the floodplain outflow nodes still require counting. Another common error is to activate a component or process switch without preparing the required data file. For example, an error will occur if MUD = 1 in the CONT.DAT file, but the data file SED.DAT is not prepared or if the sediment concentration in INFLOW.DAT is left out.

One data error that is very difficult to resolve is the array allocation violation where the array index number may be zero or larger than the assigned value. For example, the rainfall distribution variable R_DISTRIB(I) may be allocated a maximum 100 array space locations in memory (I = 100) and the number that the computer is assigning to I is 101. When this type of error is encounter, the FLO-2D model is terminated with a generic error message without indicating the file location or line entry of the error. To locate the data error, it is necessary to turn off all of the components and add them back to the model one at time until the error is encountered again. You can reset SIMUL in CONT.DAT to just after the time that the error was encountered to reduce debugging the model. If your debugging methods are ineffective, send a zipped copy of the data files to FLO-2D and described the problem in a few sentences.

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.

List of the Fifteen Most Common Data Errors
1. Missing data entries. Insufficient data was provided to the model.
2. Switches were activated without the corresponding data or files (for example, see MUD, ISED, etc., in the CONT.DAT file).
3. There was missing or additional lines in a data file when switch is activated. Observe the *** *Notes:* *** in the file descriptions.
4. Percentages were expressed as a number instead of a decimal. See the description of XCONC in CONT.DAT and the HP(I,J,3) variable in INFLOW.DAT.
5. The IDEPLT variable was improperly assigned in INFLOW.DAT for the graphics mode.
6. Corresponding grid element additions were left out of the CADPTS.DAT when additional grid elements were assigned to the FPLAIN.DAT file after it had been prepared with the GDS program. The additional grid elements were not been correctly assigned to the contiguous grid elements in the FPLAIN.DAT file.
7. Channel infiltration switch INFCHAN was not 'turned on' in the INFIL.DAT file.
8. Either or both channel and floodplain outflow elements were not assigned.
9. IQDIR channel extension directions were assigned such that the extensions intersect.
10. Channel elements that are contiguous but do not share discharge were not identified with the NOFLOC variables in CHAN.DAT file.
11. The street width exceeded the grid element side.
12. The array limitation for a variable was exceeded.
13. The available floodplain surface area was exceeded by assigning channels, streets, ARFs and/or multiple channels with too much surface area. Review the SURFAREA.OUT.
14. 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).
15. The ISEDN switch for channel sediment transport was not 'turned on' in the CHAN.DAT file for the channel segment.

The following hints are provided to assist the user that encounters runtime errors associated with a specific grid element during a FLO-2D flood simulation. If the program stops executing after several output time intervals but has not reached the prescribed simulation time, the computational error may be the result of the inappropriate elevations or n-values assigned to a grid element. Usually, the problem grid element will be listed at the end of the BASE.OUT file. The user is encouraged to review the following before revising the stability criteria or the minimum timestep:

- Review the grid element elevation and Manning's n-value for the listed grid element as well as the contiguous 8 grid elements. This can be done in the FLOENVIR program.
- Check the channel data to determine if the grid element contains a channel and then check the relationship between the channel bed and bank elevations and the floodplain elevation within the grid element.
- Review the floodplain area available within the grid element including loss of floodplain area due to channels, streets, ARFs or multiple channels.

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 create linear channel bed slopes. Mismatched channel morphology is a primary source of numerical stability problems. To compute smooth hydraulics between two channel grid elements, adjust the bed slope, cross section flow area or roughness values. Try to avoid abrupt changes in cross sections geometry from one channel element to another. The channel flow area should make a gradual transition from a wide, shallow cross section to a narrow deep cross section. This cross section transition should occur over several channel grid elements. Adjust the channel geometry so that the change in flow area between channel elements is 25% or less. Similar water surface elevations and flow depths between contiguous channel elements should have approximately the same flow area.

To address these channel problems, try the following measures:

- Increase the roughness in wide, shallow cross sections and decrease the roughness in narrow deep channel grid elements.
- Reduce the difference between the cross section areas. Avoid abrupt cross section transitions between channel elements. Adjust the channel cross section geometry in the PROFILES. Using PROFILES, you can interpolate between surveyed cross sections.
- Review and adjust the bed slope with the PROFILES program.
- Select a longer channel length within the channel grid element.

Review the following files and programs to analyze volume conservation problems: SUMMARY.OUT, CHVOLUME.OUT, CHANGRID.OUT, CHANMAX.OUT, TIME.OUT, BASE.OUT, ROUGH.OUT, CHANNEL.CHK, SURFAREA.OUT and the PROFILES program.

Troubleshooting Guide

Use the following troubleshooting guide to assist you in debugging runtime errors.

Troubleshooting Guide

Problem	Possible Cause/Solution
1. Program will not run.	a. Data errors. Reduce the number of components until the model runs. b. The executable program was damaged. Reload the program or contact technical support. c. The model is not properly licensed. Contact technical support.
2. Program stops before the first timestep or after several timesteps with data file error indicated on the screen.	a. Check the ERROR.CHK file or the data file identified by the program. b. Review the backup file (*. BAC). c. Review the List of Common Data Errors.
3. Program stops after several timesteps indicating a numerical stability error. The grid element causing the stability error is listed on the screen or at the end of the BASE.OUT file.	a. Stability criteria were not met. Review and revise the elevation and roughness data for the indicated grid element at the end of the BASE.OUT file. The ROUGH.OUT and TIME.OUT files will help to locate the problem grid element. Also, check the contiguous grid elements in the 8 directions. b. Minimum timestep is too large. Reduced DTMIN in the TOLER.DAT file. c. Increase the DEPTOL variable in TOLER.DAT file. d. For fully dynamic wave KMODEL = 3, reduce the WAVECEL, WAVEMIN, and WAVEMAX variables in the TOLER.DAT file.
4. Volume was not conserved or there is discharge surging. Surging is usually associated with channel flow. The volume conservation may indicate either a loss or gain of volume. A review of the CHANMAX.OUT, HYCHAN.OUT, BASE.OUT, CHVOLUME.OUT, CHANSTABILTY.OUT, TIME.OUT, and SUMMARY.OUT files may also indicate that the discharges are surging (alternating from low to high discharges). Floodplain surging can occur but is uncommon. Generally, floodplain can on flat or mild slopes where the flow is concentrated into a row or column of grid elements like a rectangular channel. NOTE: Channel surging may occur and still meet volume conservation.	The surging may be related to numerical instability, abrupt changes in channel geometry, inappropriate slopes, supercritical flow or variable mudflow sediment concentrations. Mismatched slope, flow area and n-values are the most common causes of channel instability. A combination of methods can fix instability. Numerical Instability: a. Abrupt changes in slope or severe adverse slope may cause instability. Use the PROFILES program to fix irregular bed slope conditions. b. 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. c. Decrease the WAVECEL, WAVEMIN, WAVEMAX and DEPTOL variables in the TOLER.DAT file. Suggested minimum values: WAVECEL = 0.1; WAVEMIN = 0.2; WAVEMAX = 1.0; DEPTOLCH = 0.10.

Troubleshooting Guide	
Problem	Possible Cause/Solution
	d. Insufficient floodplain area. Small floodplain surfaces in channel grid elements can exacerbate unsteady flow. e. Increase the n-values for the grid elements in the vicinity of the surging flow. f. Increase the channel length within the grid element. g. The hydraulic structure discharge rating curve or table are poorly matched with the upstream or downstream channel hydraulics. Review the hydraulic structure rating curve or table. Supercritical Flow: a. Adjust the limiting Froude number using the FROUDL variable in the CONT.DAT file or the FROUDC variable in the CHAN.DAT. b. Increase the floodplain or channel roughness values. c. Modify the slope. The grid elevations assigned by the GDS or CADD software 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: a. Review the sediment concentration in the inflow hydrographs in FPINOUT.DAT or CINOUT.DAT files. See also Problem #9 below.
5. 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 exceeding the numerical stability criteria. The change in flow depth for a timestep is too large. One of primary reasons for a slow flood simulation is that the relationship between the discharge flux and grid element surface area is poor. The rate of change in the discharge is too high for the select grid element size. It is recommended that $Q_{peak}A_{surf} << 1.0$ Increasing the grid element size is the best way to fix a very slow model. Other solutions may include: a. Increase the grid element size or the channel size. b. Create a more uniform channel or floodplain slope. c. Reduce the discharge flux by reducing the rate of rise or steepness of the hydrograph. d. Revise the roughness values or limiting the supercritical flow. e. Reduce the channel width or other parameters to increase the floodplain surface area. Review the SURFAREA.OUT file. f. Review the solutions to Problem #4.

Troubleshooting Guide	
Problem	Possible Cause/Solution
6. Graphics display does not activate.	The graphics display is controlled by four variables: LGPLOT, GRAPTIM and INPLOT found in the CONT.DAT file and IDEPLT in INFLOW.DAT. IDEPLT is the floodplain or channel inflow grid elements whose hydrograph will be plotted on the display. a. Set LGPLOT = 2. b. Check the GRAPTIM variable for an acceptable update interval. c. Set INPLOT switch to appropriate value 0,1 or 2. d. Make sure that an inflow hydrograph assigned to a grid or channel element in INFLOW.DAT matches the IDEPLT variable.
7. The inflow hydrograph does not plot in the graphics display.	a. No hydrograph is associated with the IDEPLT variable. b. The hydrograph duration is too long. Reduce the hydrograph length. c. The rainfall duration is too long. Reduce the rainfall time. d. Inappropriate peak discharge or total rainfall values distort the scale for hydrograph plot. Consult technical support.
8. Program stops. Flow depths exceed than 100 ft.	Flow depths greater than 100 ft cause the FLO-2D program to terminate. If flow depths greater than 100 ft are expected consult technical support to revise the model. If depths of 100 ft are not expected, then excessive ponding or surging is occurring. Check the end of the BASE.OUT file to identify the problem grid. a. Check the elevation of the problem grid element in the FPLAIN.DAT. b. If surging is causing the high flow depth see Problem #4 above.
9. Erratic discharge in the channel elements. A review of plotted hydrographs in HYDROG or an examination of the CHANMAX.OUT or HYCHAN.OUT files will reveal if the flow discharge between contiguous channel elements is surging with spikes when a consistent rise or fall of the downstream discharge is expected.	Channel surging can be natural phenomena. Rivers rise and fall over a few tenths of a foot in matter of seconds. During high flow, the variation in discharge associated with stage change~0.2 ft can be 1,000 cfs or more. If no channel surging is expected, there are several solutions: a. Review the solutions to Problem #4 above. b. Channel grid elements that are contiguous but do not share discharge must be identified in the CHAN.DAT file. NOFLOC variables in the CHAN.DAT may not be properly assigned. See the CONFLUENCE.OUT file. c. The NOFLO1 or NOFLO2 grid elements were mis-identified. d. The channel grid elements in the CHAN.DAT file (ICHANGRID) in line 3 are mislabeled. e. Review Item 4 solution above.

Troubleshooting Guide	
Problem	Possible Cause/Solution
10. Erratic flow in the floodplain grid elements.	Erratic flow in the floodplain grid elements is usually the result of errors in the FPLAIN.DAT file. This type of error generally occurs when the user edits the FPLAIN.DAT file manually and adds, subtracts or moves grid elements around. On occasion, a flood simulation may confine overland flow in a row or column of flat or mild slope floodplain grid elements. This flow could respond just like surging channel flow. To fix this problem, impose some slope to the row or column of grid elements.
11. Channel extends into another channel element. If the width is too large (width exceeds 95% of the side of a grid element) or there is no floodplain surface left, the model will automatically extend the channel into a neighboring grid element. If the extension intersects another channel element or extension the model will stop. Review the ERROR.CHK and CHANNEL.CHK files.	The variable IQDIR must be specified in the CHAN.DAT file. If the channel extension intersects another channel, there are several choices: a. Change the IQDIR direction. You can visualize the channel extensions in the FLOENVIR. Modify other channel extensions and the channel position to eliminate the channel extension intersections. It may necessary to extend the channel in the opposite direction for one or two elements. b. Reduce the channel extension by decreasing the channel top width or the channel length (XLEN). If it is necessary to reduce the channel width, identify which channel geometry is assigned to the problem grid element, rectangular, natural, trapezoidal or cross section. Then reduce the appropriate parameters in the CHAN.DAT file. c. If the extension is related to the lack of surface area in channel bank floodplain element, decrease or eliminate the other component attributes that may be reducing the surface area such as streets, ARFs or multiple channels.
12. Program stops prior to first timestep identifying one or more grid elements with too little floodplain surface.	Generally, this should not be a concern for channel elements because the model will automatically assign more elements to the channel extension if there is too little floodplain. If this problem occurs and the floodplain surface is less than 5%, then there are several solutions: a. Reduce the ARF value, multiple channel area or street area. b. 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. c. As a last resort the grid element size can be increased, but this requires the re-generation of the grid system.
13. If grid elements are added or deleted to grid system and the FPLAIN.DAT file is revised, the CADPTS.DAT will report errors if it is not also modified.	The CADPTS.DAT file must be changed to reflect the revisions in the FPLAIN.DAT file.

Appendix

Obsolete Pre-Processor Programs

There are four processor programs CADATA, CHECKER, POWER, XSEC that are no longer required by FLO-2D. These may valuable for a unique application.

CADATA

CADATA is a preprocessor program that reformats the topographic grid point data exported from the CADD program DTM into a data file used by FLO-2D. The data exported from the CADD program should have the following format separated by spaces:

Grid Point No. X-Coordinate Y-Coordinate Elevation Descriptor

The descriptor (DESC) is an optional value that can be used to spatially vary the Manning's n-value (roughness coefficient) in the CADD drawing. It may be left blank and a generic n-value can be assigned when the CADATA program is run.

The CADATA program reorganizes the topographic data into a file format that identifies contiguous elements in the four directions. The program is initiated by typing or double clicking on CADATA. CADATA is user-interactive and requests the following information:

- Name of data file;
- Orientation of the grid elements and the coordinate system;
- n-values are that are either specified in CADD file (as DESC) or will globally assigned by CADATA;
- Requirement to renumber the grid elements starting with 1;
- Number of grid elements;
- Size of the grid elements;
- Generic n-value for all grid elements.

The output of the CADATA program is a data file named FPLAIN.DAT in the following format with the data separated by a space:

Grid #	Node to the North	Node to the East	Node to the South	Node to the West	Manning's n-value	Elevation

The program will evaluate the distance between the grid elements from the x- and y-coordinate values within an accuracy tolerance of one foot. The spacing does not have to be exact. No further modification of this file is necessary unless elevations or n-values are changed.

The x- and y-coordinates for each grid element are retained in the CADPTS.DAT file. This file is accessed by FLO-2D when writing output files. For example, to plot maximum flow depth contours in the original CADD drawing, FLO-2D writes the x- and y-coordinates from the CADPTS.DAT file to the DEPTH.OUT file. *Any changes in the number or location of the grid elements in the FPLAIN.DAT file must be matched by corresponding changes in the CADPTS.DAT file.*

CHECKER

CHECKER is an older DOS based program that will check whether the grid elements are properly correlated in the FPLAIN.DAT file. It is not necessary to use this program if the FPLAIN.DAT was developed with either the GDS or CADATA programs. CHECKER would only be useful if the FPLAIN.DAT file has been modified after its creation. It reports those grid elements whose neighbors are not correctly correlated to file. For example, if grid element 423 has grid element 567 to the north, then grid element 567 must have grid element 423 to the south. If this cross correlation in the FPLAIN.DAT file is in error, then CHECKER will report the errors to the CHECK.OUT file. To run CHECKER, simply copy the CHECKER.EXE file into the project subdirectory and double click on the file name in Explorer. The CHECK.OUT file will also contain information regarding floodplain depressions. This depression information is obsolete and should be ignored.

POWER

The POWER program is a simple program which computes a power regression relationship ($y = a\ x^b$) based on a least squares fit for any x and y variable data set. POWER can be used to generate the hydraulic structure rating curve from a set of depth (x) versus discharge (y) data. It computes a set of regression statistics including as correlation coefficient and standard error of estimate. The program will ask for an input and output file name at runtime. The POWER data format is as follows with the ASCII data separated by a space:

Line 1: Number of data sets (no limit).
Line 2: Descriptive name for the data set (limit 10 characters).
Line 3: Number of data pairs (limit 300 coordinates).
Line 4 to the end of the data set: x-data, y-data (Line 4 is repeated for each data pair).

XSEC

There are four channel geometry options: rectangular channels, trapezoidal channels, variable area channel geometry relationships and cross sections. XSEC computes the variable area channel geometry relationships of flow area, top width, and wetted perimeter as a power function of the flow depth. The variable area channel geometry option uses the cross section survey data to determine the channel geometry as a power function of the depth. This option should be used when the cross section data is incomplete or poorly defined. The cross section flow area A is given by:

$$A = a\ d^b$$

where (a) is a regression coefficient and (b) is a regression exponent. The top width and wetted perimeter relationships as a function of the depth have a similar form and are also required by the channel routing routine. These cross section power function relationships create an accurate and computer efficient estimate of the cross section geometry. This data is input into the CHAN.DAT file. A set of regression statistics are computed in the output file.

The program will interactively request the input and output file names. Metric units can be specified. The output file CHANXSEC.OUT lists the thalweg depth, average depth for lower 50% of the channel and channel geometry in a format that can be imported to the CHAN.DAT file. The XSEC input file must be created in the following format with the data in columns separated by a space:

Line 1: Number of cross sections (no limit).
Line 2: Cross section ID (limit 10 characters).
Line 3: Number of coordinate points in each cross section (limit 300 coordinates).
Line 4: x-coordinate, y-coordinate from left bank looking downstream (repeated).

Note: Line 4 is repeated for the number of coordinate points in each cross section. Lines 2 thru 4 are repeated for each cross section. A partial example of a XSEC program data file follows:

The cross section data option should be used in the FLO-2D model (SHAPE = 'N' option in CHAN.DAT) whenever possible. The cross section data is represented in the model as a rating table and although its application consumes more computer resources in terms of memory and processor time, it is more accurate. The channel geometry regression relationship are less accurate when the channel is more U-shaped (narrow and deep), when there is a limited amount of cross section data or when there is a significant variation is the cross section geometry.