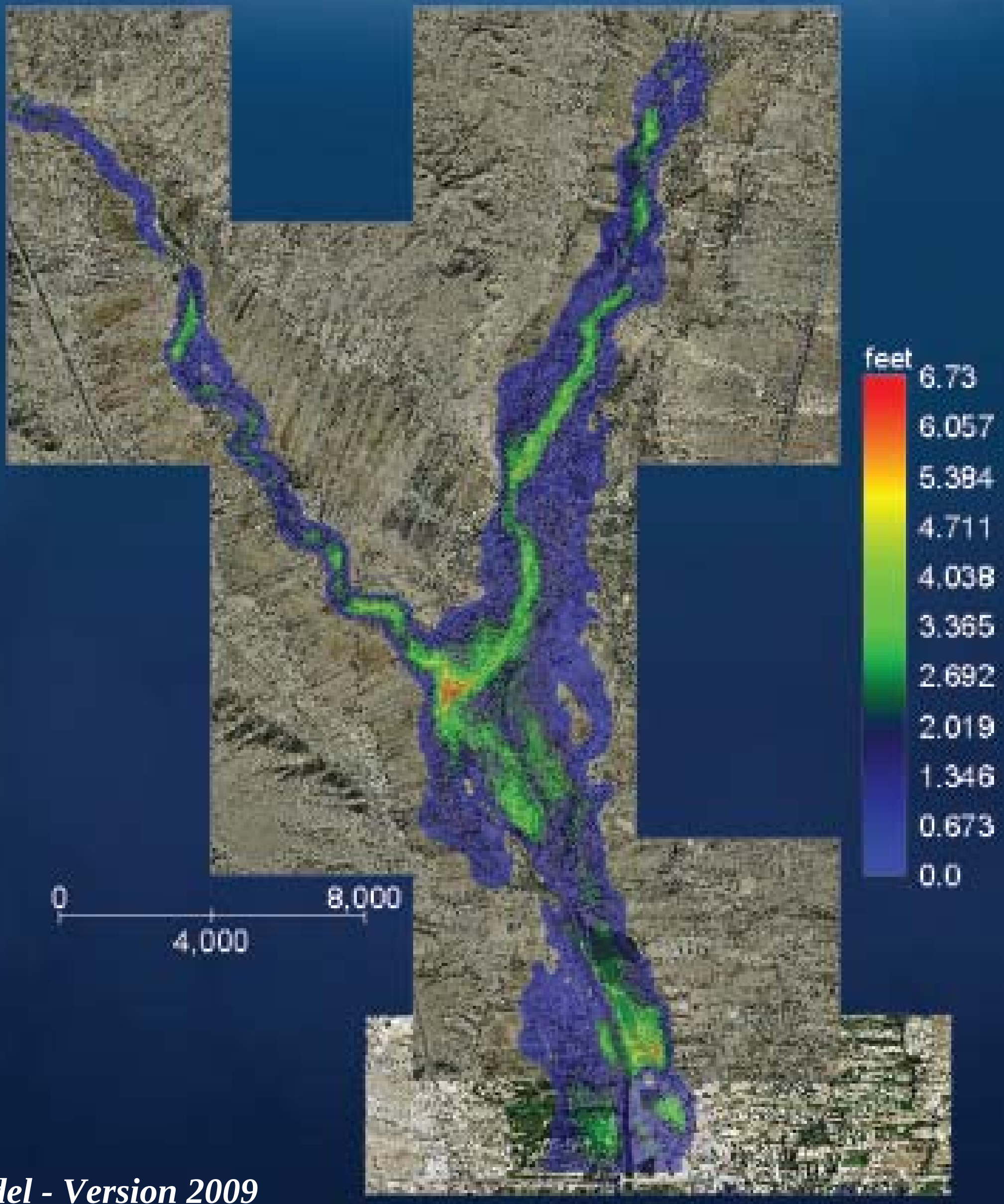
FLO-2D

REFERENCE MANUAL



Basic Model - Version 2009

A few comments on modeling free surface flows…

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

Reliable flood hazard delineation requires a critical review of model applicability, modeling assumptions, and the available data bases. While 2-D models have become more versatile with increasing computer resources, inadequate hydrographic data bases still limit the accuracy of flood hazard delineation. Digital terrain models are becoming the foundation of high resolution mapping, but post-flood event surveys of high water marks and aerial photography of the area of inundation are either unavailable or perhaps were collected long after the flood waters have receded. Correlating the area of inundation with flood peak discharge can lead to the harsh realization that our best discharge measurements or gaging data have limited accuracy at high flows. Our modeling and mapping results may be only as good as the model calibration to post-flood data.

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

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

BRIEF OVERVIEW

FLO-2D is a volume conservation flood routing model. It is a valuable tool for delineating flood hazards, regulating floodplain zoning or designing flood mitigation. The FLO-2D Basic Model will simulate river overbank flows, but it can also be used on unconventional flooding problems such as unconfined flows over complex alluvial fan topography and roughness, split channel flows, and urban flooding. The FLO-2D Basic Model is on FEMA's list of approved hydraulic models. The FLO Pro Model expands the basic model to include the additional components: sediment transport, mud and debris flows, storm drains, dam and levee breach, and groundwater.

The FLO-2D software package includes a grid developer system (GDS) and a Mapper program that automates flood hazard delineation. The GDS will filter DTM points, interpolate the DTM data and assign elevations to grid elements. The MAPPER program automates flood hazard delineation. MAPPER will generate very detailed flood inundation color contour maps and shape files. It will also replay flood animations and generate flood damage and risk maps.

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

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

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

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

TABLE OF CONTENTS

Page

BRIEF OVERVIEW ii
LIST OF FIGURES iv
LIST OF TABLES iv

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

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

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

IV. MODEL COMPONENTS 21
  4.1 Model Features 21
  4.2 Overland Flow 21
  4.3 Channel Flow 25
  4.4 Channel-Floodplain Interface 27
  4.5 Limiting Froude Numbers 27
  4.6 Levees 28
  4.7 Levee and Dam Breach Failures 30
  4.8 Hydraulic Structures 31
  4.9 Street Flow 32
  4.10 Floodplain Surface Storage Area Modification and Flow Obstruction 32
  4.11 Rainfall and Runoff 33
  4.12 Infiltration and Abstraction 35
  4.13 Evaporation 37
  4.14 Overland Multiple Channel Flow 38
  4.15 Sediment Transport – Total Load 38
  4.16 Mud and Debris Flow Simulation 39

V. FLO-2D APPLICATIONS AND METHODS 41
  5.1 River Applications 41
  5.2 Unconfined Overland and Alluvial Fan Flooding 42
  5.3 Model Results – What Constitutes a Successful Flood Simulation? 43

VI. FLO-2D MODEL VALIDATION 44

VII. REFERENCES 45

LIST OF FIGURES

Page
Figure 1. Physical Processes Simulated by FLO-2D 3
Figure 2. Channel – Floodplain Interface 4
Figure 3. FLO-2D Flow Chart 6
Figure 4. Discharge Flux across Grid Element Boundaries 12
Figure 5. FLO-2D Stability Criteria Flow Chart....13
Figure 6. Overland Tsunami Wave Progression in an Urban Area (Waikiki Beach, Hawaii) 18
Figure 7. Overland Flow Routing Subroutine Flow Chart....24
Figure 8. Channel Extension over Several Grid Elements....25
Figure 9. Levees are depicted in Red and the River in Blue in the GDS Program....29
Figure 10. Levee Freeboard Deficit Plot in Mapper 29
Figure 11. Example of Levee Breach Urban Flooding 30
Figure 12. Example of a Proposed Domestic Water Supply Reservoir Breach Failure 31
Figure 13. Streets Depicted in Green in the FLOENVIR Program. 32
Figure 14. Area and Width Reduction Factors 33
Figure 15. Flooding Replicated from NEXRAD Data near Tucson, Arizona....34
Figure 16. Classification of Hyperconcentrated Sediment Flows 39
Figure 17. Middle Rio Grande and Rio Chama Confluence Model....41
Figure 18. Unconfined Alluvial Fan Flooding....42
Figure 19. Urban flooding with Street Flow and Building Obstruction....42
Figure 20. FLO-2D versus USGS Measured Gage Data 44

LIST OF TABLES

Table 1. Overland Flow Manning's n Roughness Values[1] 22
Table 2. Initial Abstraction 35
Table 3. Green Ampt Infiltration - Hydraulic Conductivity and Porosity....36
Table 4. Green Ampt Infiltration - Soil Suction....36
Table 5. Green Ampt Infiltration -Volumetric Moisture Deficiency 37
Table 6. Mudflow Behavior as a Function of Sediment Concentration 40

FLO-2D

Two Dimensional Flood Routing Model

I. INTRODUCTION

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

1.1 Evolution of the FLO-2D Model

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

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

FLO-2D is a physical process model that routes rainfall-runoff and flood hydrographs over unconfined flow surfaces or in channels using the dynamic wave approximation to the momentum equation. It has a number of components to simulate street flow, buildings and obstructions, sediment transport, spatially variable rainfall and infiltration, floodways and many other flooding details. Predicted flow depth and velocity between the grid elements represent average hydraulic flow conditions computed for a small timestep (on the order of seconds). Typical applications have grid elements that range from 10 ft to 300 ft on a side and the number of grid elements is unlimited.

1.2 Modeling the Hydrologic System with FLO-2D

The FLO-2D system consists of processor programs to facilitate graphical editing and mapping and components that simulation channel and floodplain detail. The Grid Developer System (GDS) generates a grid system that represents the topography as a series of small tiles. The FLO-2D model has components for rainfall, channel flow, overland flow, street flow, infiltration, levees and other physical features. The GDS program is used to spatially edit the grid system attributes. The PROFILES program edits channel slope and cross section shape. Flood routing results can be viewed graphically in the MAXPLOT, MAPPER and HYDROG (plots 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 either as a floodplain or channel inflow. As the floodwave moves over the floodplain or down channels or streets, flow over adverse slopes, floodwave attenuation, ponding and backwater effects can be simulated. In urban areas, buildings and flow obstructions can be simulated to account for the loss of storage and redirection of the flow path. The levee component can be used to test mitigation alternatives.

Channel flow is one-dimensional with the channel geometry represented by either by natural, rectangular or trapezoidal cross sections. Street flow is modeled as a rectangular channel. Overland flow is modeled two-dimensionally as either sheet flow or flow in multiple channels (rills and gullies). Channel overbank flow is computed when the channel capacity is exceeded. An interface routine calculates the channel to floodplain flow exchange including return flow to the channel. Similarly, the interface routine also calculates flow exchange between the streets and overland areas within a grid element (Figure 2). Once the flow overtops the channel, it will disperse to other overland grid elements based on topography, roughness and obstructions.

It is important to assess the level of detail required on a given project. FLO-2D users have a tendency to put more detail into their models than is necessary for a large flood event. Preparation of channel flow, street flow, buildings and flow obstructions data files can be time consuming and should be tailored to meet the project needs. The desired accuracy of predicted water surface elevations should be consistent with the resolution of the mapping, survey and hydrologic data bases. Simulating large floods requires less detail than shallow flood or mitigation design models.

The FLO-2D Basic Model does not have several components that are available in the Pro Model. The Pro Model has the following components not found in the Basic Model:

- Sediment transport
- Mud and debris flows
- Dam and levee breach
- Groundwater
- Storm drains

If your project requires one or more of these components please upgrade to the Pro Model.

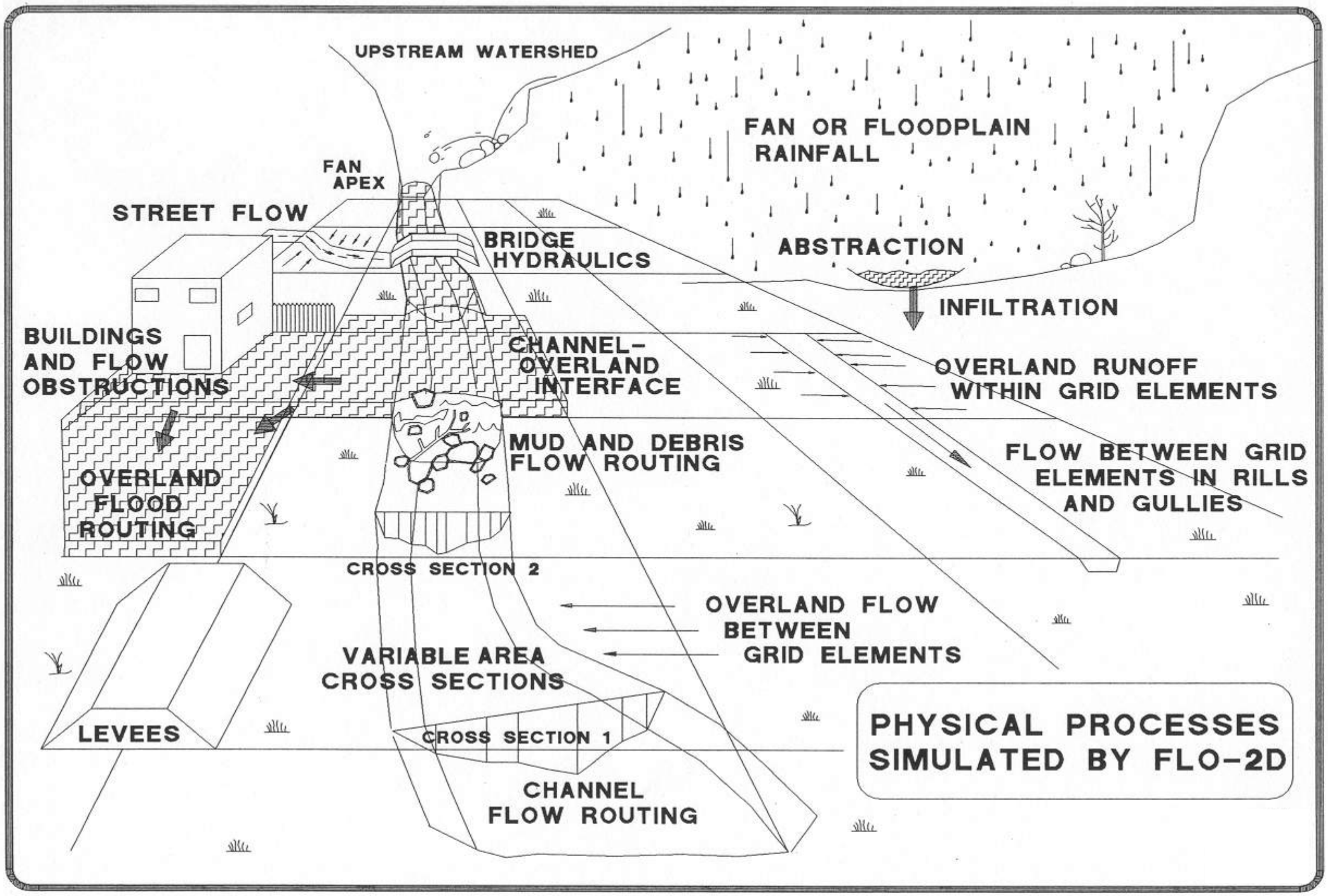


Figure 1. Physical Processes Simulated by FLO-2D

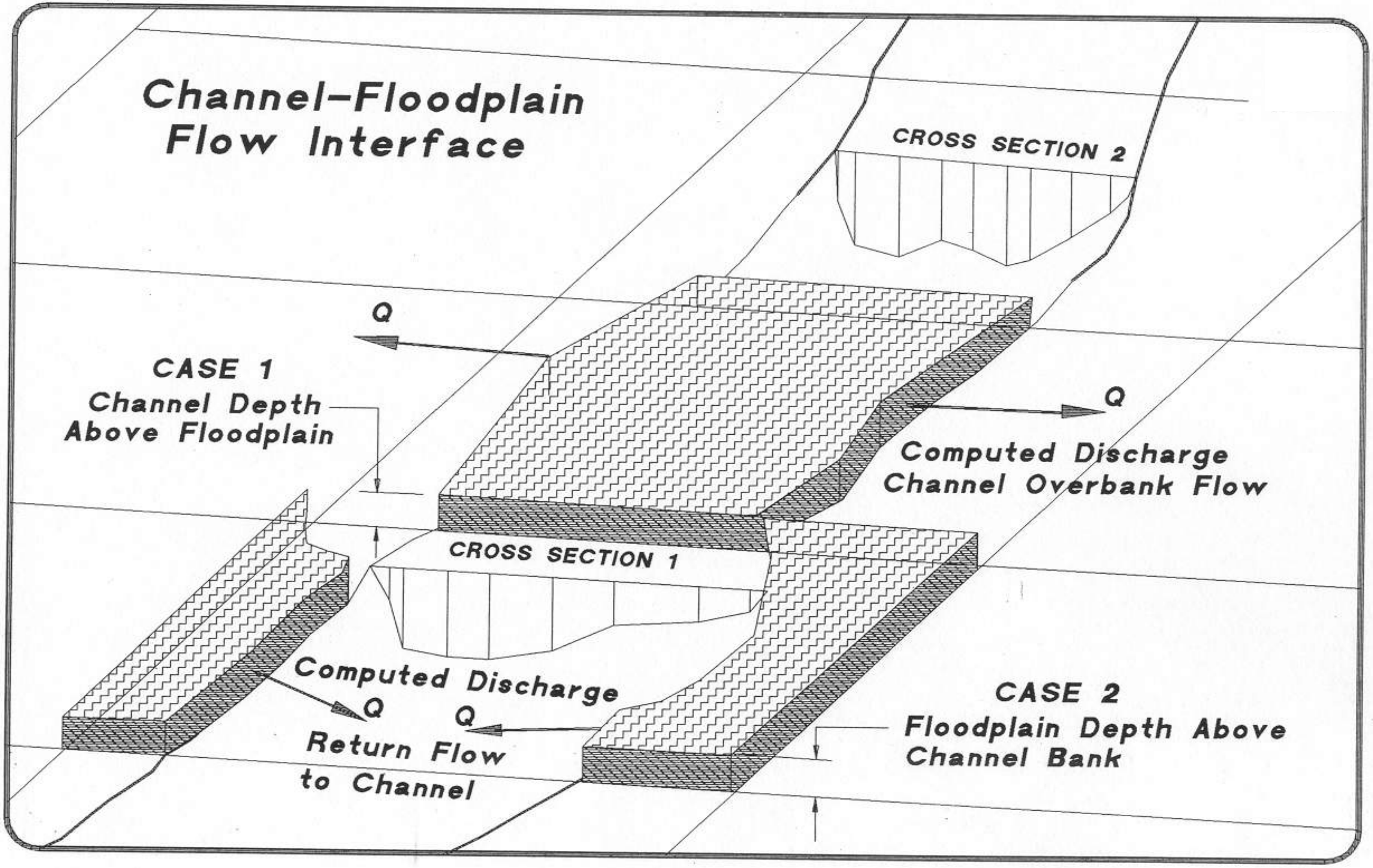


Figure 2. Channel – Floodplain Interface

1.3 Getting Started on a Project – A Brief Overview

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

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

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

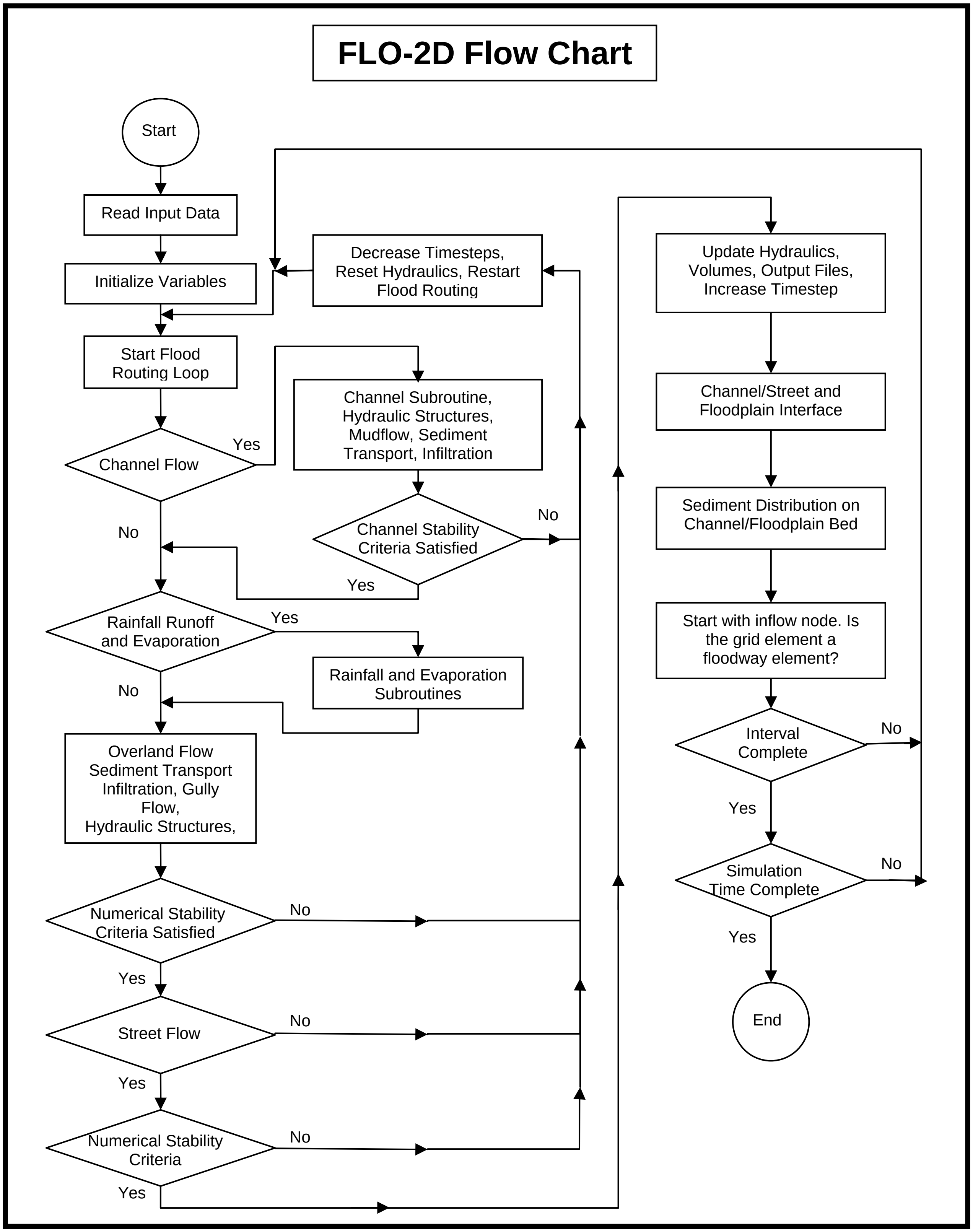


Figure 3. FLO-2D Flow Chart

II. FLO-2D MODEL THEORY

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

2.1 Governing Equations

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

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

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

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

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

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

This illustrates that the application of the kinematic wave ($S_o = S_f$) on moderately steep slopes with relatively steady, uniform flow is sufficient to model floodwave progression and the contribution of the pressure gradient and the acceleration terms can be neglected. The addition of the pressure gradient term to create the diffusive wave equation will enhance overland flow simulation with complex topography. The diffusive wave equation with the pressure gradient is required for floodwave attenuation and change in storage on the floodplain. The local and convective acceleration terms are important to the flood routing for flat or adverse slopes or very steep slopes or unsteady flow conditions. Only the full dynamic wave equation is applied in FLO-2D model.

2.2 Solution Algorithm - How the Model Works

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

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

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

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

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

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

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

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

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

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

To summarize, the solution algorithm incorporates the following steps:

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

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

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

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

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

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

   timestep Δt between time i and i + 1.
7. The numerical stability criteria are then checked for the new grid element flow depth. If any of the stability criteria are exceeded, the simulation time is reset to the previous simulation time, the timestep increment is reduced, all the previous timestep computations are discarded and the velocity computations begin again.
8. The simulation progresses with increasing timesteps until the stability criteria are exceeded.

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

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

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

where:

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

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

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

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

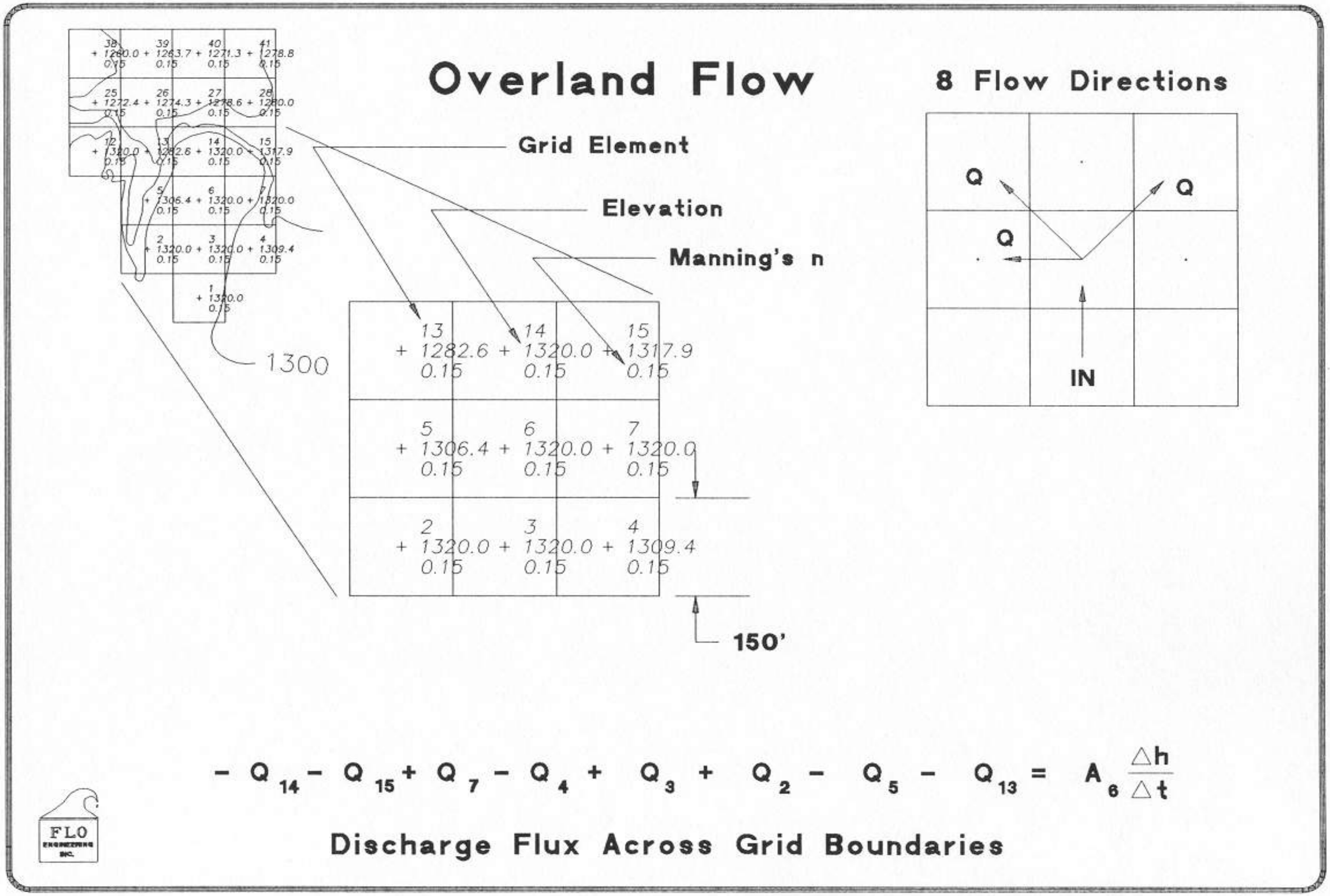
where q_o is the unit discharge, S_o is the bed slope and ζ is an empirical coefficient. The coefficient ζ was created as a variable unique to the grid element and is adjusted by the model during runtime within a minimum and maximum range set by the user. Similar to the CFS criteria, when this numerical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

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

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

2.3 The Importance of Volume Conservation

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



Overland Flow

$$-Q_{14}-Q_{15}+Q_7-Q_4+Q_3+Q_2-Q_5-Q_{13}=A_6\frac{\triangle h}{\triangle t}$$

Discharge Flux Across Grid Boundaries

Figure 4. Discharge Flux across Grid Element Boundaries

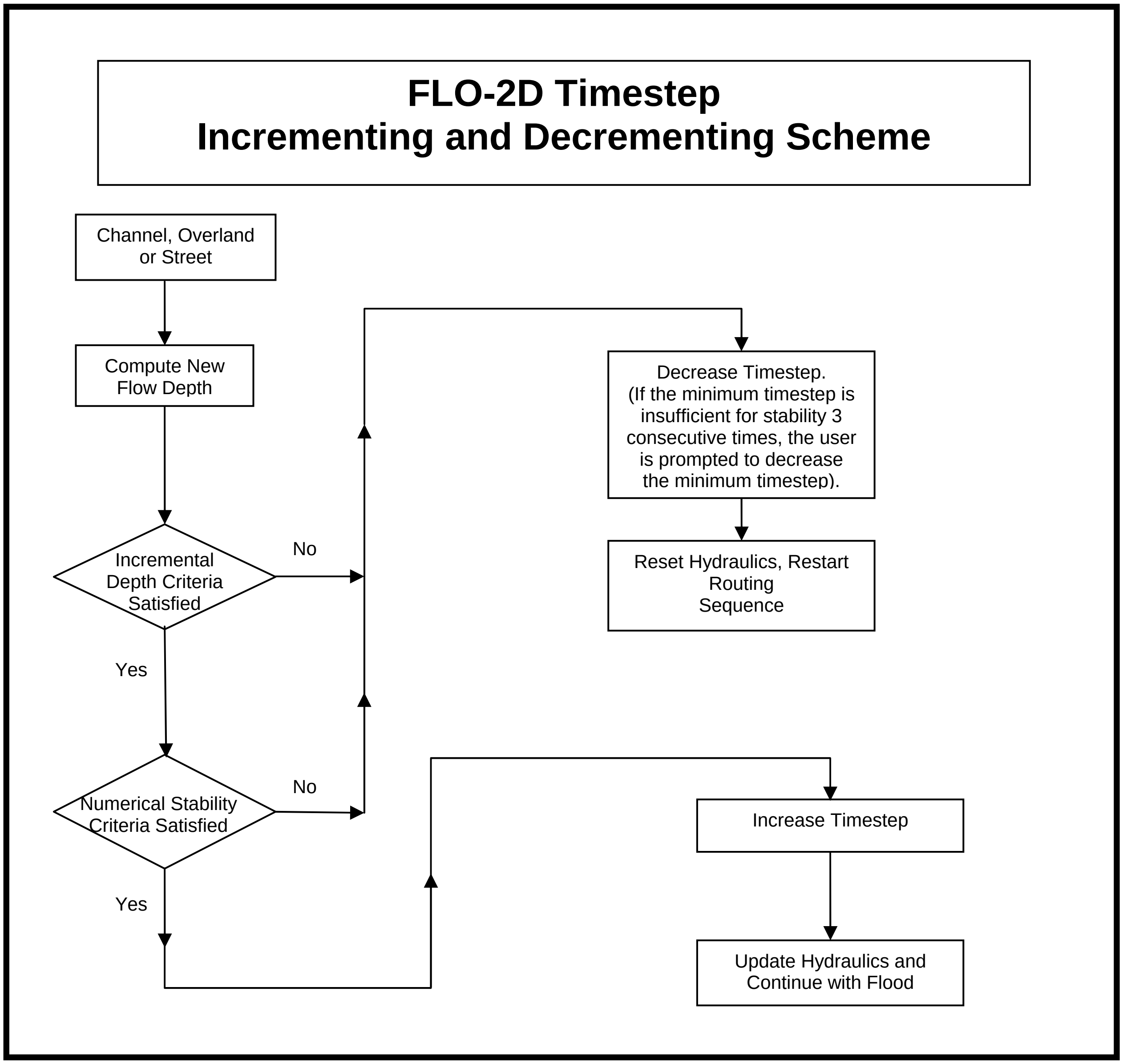


Figure 5. FLO-2D Stability Criteria Flow Chart

III. FLO-2D MODEL SYSTEM

3.1 Assumptions

Conceptualization

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

Spatial Resolution

The spatial and temporal resolution of the FLO-2D model is dependent on the size of the grid elements and rate of rise in the hydrograph (discharge flux). The rate of change in flood discharge results in an incremental change in the flow depth when distributed over the available grid element surface area for a given timestep. Smaller grid elements may improve the resolution of the flood distribution at the cost of increased computational time, more extensive data files and boundary conditions. A balance must be struck between the number of grid elements and an acceptable computational time. A grid size of 10 ft (3 m) to 300 ft (100 m) is usually appropriate for most simulations. Smaller grid elements will not only significantly increase the number of grid elements (the number of grid elements is quadrupled each time the grid element size is divided by two), but the rate of discharge flux per unit area of the grid element increases.

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

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

Floodwave Attenuation and Discontinuities

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

Simulating Ponded Water Conditions

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

Basic Assumptions

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

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

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

Rigid Bed Model

Sediment transport is not simulated in the FLO-2D Basic Model and a rigid bed is presumed for the flood simulation. Rigid boundary conditions are appropriate for flow over steep slopes, urban flooding and mudflow events. The area of inundation associated with extreme flood events are generally unaffected by bed changes. Channel bed changes generally deviate about a mean condition, and the portion of the flood volume stored in the channel can be small relative the volume on the floodplain. It is assumed in rigid bed simulations that the average flow hydraulics and water surface are not appreciably affected by the scour and deposition that might occur in an individual grid element. FEMA Flood Insurance Studies require that flood simulations are rigid bed models.

Simulating a mobile bed can be more important for smaller floods, for alluvial fan flows and where channel avulsion or sediment deposition might change the flow path. The FLO-2D Pro Model can simulate sediment transport and a mobile bed.

3.2 Parameter Variability

Roughness Adjustments

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

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

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

where:

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

Ponded Water Conditions

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

Flow Contraction and Expansion

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

Limiting Froude Numbers

Limiting Froude numbers can be specified for overland flow, channel flow and street flow. As an introduction, limiting Froude numbers can be used to adjust the relationship between the flow area, slope and n-values. When the computed Froude number exceeds the limiting Froude number, the n-value is increased for that grid element by a small incremental value for the next timestep. In this manner, the flow can be forced to be subcritical if in reality, critical or supercritical flow is not possible. For example, in steep-slope sand bed channels, high energy flows may entrain more sediment to sustain subcritical flow. In this case, the limiting Froude number might be set to 0.9. For flow down steep streets, a maximum Froude number of 1.2 to 1.5 may be specified to limit the supercritical flow. Since FLO-2D does not simulate hydraulic jumps, the limiting Froude number should represent average flow conditions in a channel reach. During the falling limb of the hydrograph when the Froude decreases to a value less than 0.5, the flow resistance n-value decreases by a small incremental value until the original n-value is reached. The limiting Froude number will be discussed in more detail in Section 4.6.

Flood Parameter Variability

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

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

3.3 Inflow and Outflow Control

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

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

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

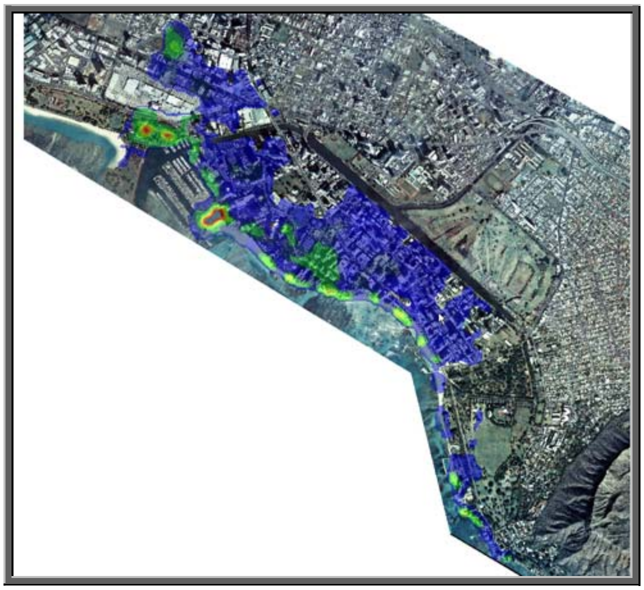


Figure 6. Overland Tsunami Wave Progression in an Urban Area (Waikiki Beach, Hawaii)

3.4 Floodplain Cross Sections

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

3.5 Grid Developer System (GDS)

The Grid Developer System (GDS) pre-processor program creates and edits the FLO-2D grid system attributes and data files and provides a platform for running the other pre- and post-processor programs. The GDS will overlay the grid system on the DTM points, interpolate and assign elevations to the grid elements. The GDS will then automatically prepare the basic input files for the FLO-2D model. Geo-referenced aerial photos, shape file images or maps can be imported as background images to support the graphical editing. In addition to developing the FLO-2D grid system, the GDS also provides important editorial features including the assignment of spatially variable attributes such channels, levees, streets, infiltration, area and width reduction factors, floodplain elevation and roughness, inflow and outflow nodes and rill and gully geometry. It allows selection of individual elements or large groups of node using the mouse. Rainfall can also be spatially varied. Detailed instructions are presented in the GDS Manual. There is also a Graphical User Interface (GUI) program for the FLO Basic Model. This GUI program is considered to be obsolete and will not be part of the FLO Pro Model system.

3.6 Graphical Output Options

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

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

MAPPPER is the primary program for displaying the FLO-2D results. It can create high resolution color contour plots. Several map combinations can be created: grid element or DTM point plots, line contour maps and shaded contour maps. Maps can be created for ground surface elevations, maximum water surface elevations, maximum floodplain flow depths, maximum velocities, maximum static and dynamic pressure, specific energy, and floodway delineation. One of the most important features of MAPPER is its capability to create flood depth plots using the DTM topographic points. When the user activates the feature, MAPPER will subtract each DTM ground point elevation from the grid element floodplain water surface elevation. The resultant DTM point flow depths can then be interpolated and plotted as color contours. Some of the MAPPER features include:

- Multiple geo-referenced aerial photos in various graphic formats can be imported such as TIFF, BMP, JPG, etc.
- Multiple layer capability including control of layer properties is available.
- Cut and view flow depth and topography profiles.

- Flood damage assessment component to compute the flood damage as function of the FLO-2D predicted maximum depths, building shape files and building value tables (dbf file).
- Flood animation. The floodwave progression over the grid system can be viewed.
- Flood maximum deposition and scour can be plotted.
- Maximum flow velocity vectors can be viewed.
- Hazard maps based on flood intensity and frequency can be created.
- GIS shape files (*.shp) are automatically created with any plotted results. These shape files can be then be imported into GIS programs.
- FEMA Digital Flood Insurance Rate Map (DFIRM) optional tool.

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

3.7 Data Output Options

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

IV. MODEL COMPONENTS

4.1 Model Features

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

- Floodwave attenuation can be analyzed with hydrograph routing.
- Overland flow on unconfined surfaces is modeled in eight directions.
- Floodplain flows can be simulated over complex topography and roughness including split flow, shallow flow and flow in multiple channels.
- Channel, street and overland flow and the flow exchange between them can be simulated.
- Channel flow is routed with either a rectangular or trapezoidal geometry or natural cross section data.
- Streets are modeled as shallow rectangular channels.
- The flow regime can vary between subcritical and supercritical.
- Flow over adverse slopes and backwater effects can be simulated.
- Rainfall, infiltration losses and runoff on the alluvial fan or floodplain can be modeled.
- 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.

Sediment transport, mudflow, groundwater, dam and levee breach, and storm drain simulation components are not available in the FLO Basic Model. These components are part of the FLO Pro Model.

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

4.2 Overland Flow

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

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

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

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

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

Some FLO-2D projects have been model grid elements inside of the channel. In this case, the channel component is not used and instead the FLO-2D grid system is draped over the channel portion of the topography. While these projects have been conducted with some success, there are several modeling concerns that should be addressed. The FLO-2D model was developed to be able to exchange 1-D channel overbank discharge with the floodplain grid elements. For this reason, the model works well on large flood events and large grid elements. When small grid elements are used inside of a channel with confined flow and large discharges and flow depths, the model may run slow. In addition, there will be zero water surface slope between some grid elements. It should be noted that the application of the

Manning's equation for uniform channel to compute the friction slope is no longer valid as the velocity approaches zero (ponded flow condition). The resulting water surface elevations can be accurately predicted but will display some small variation across the channel.

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

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

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

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

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

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

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

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

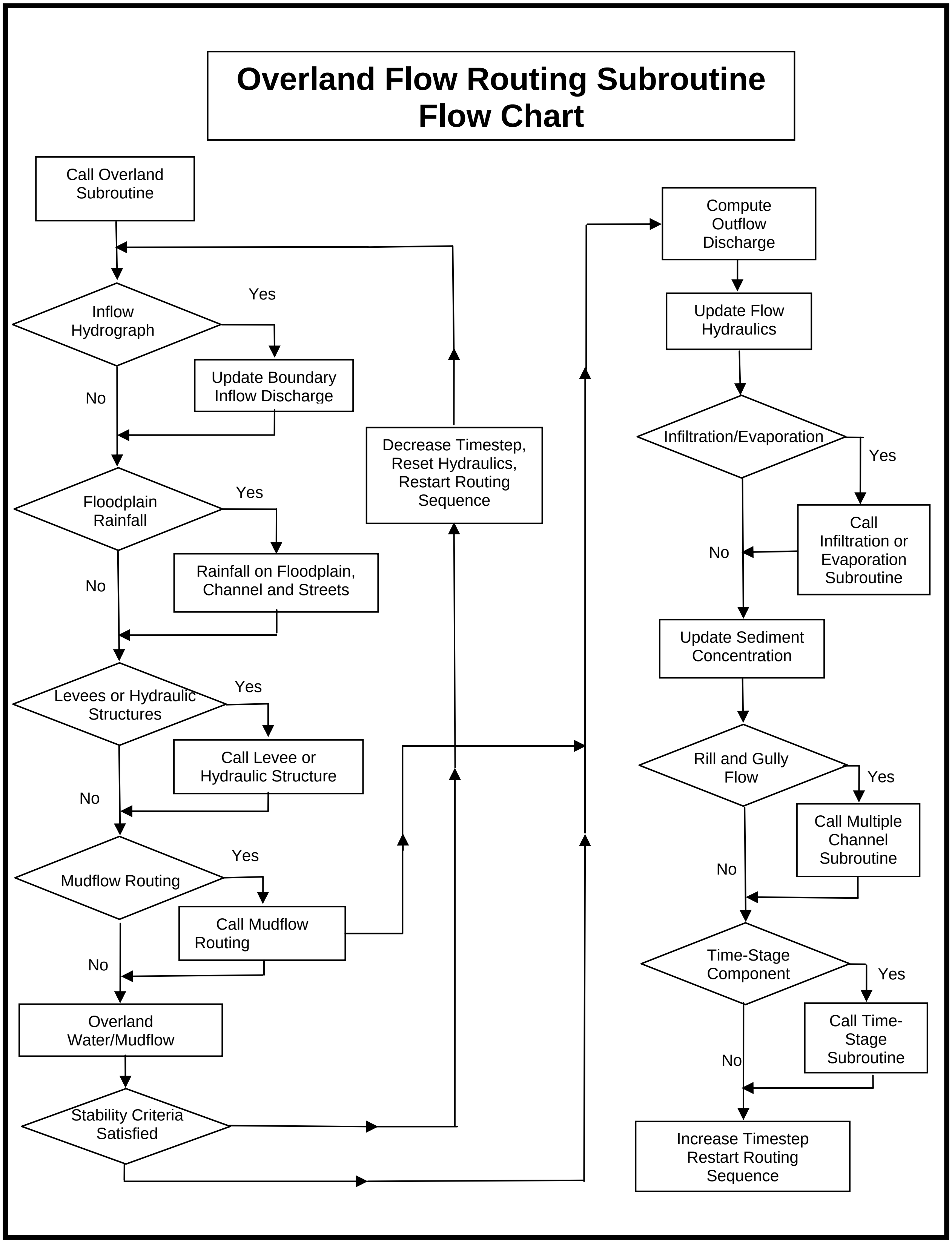


Figure 7. Overland Flow Routing Subroutine Flow Chart

4.3 Channel Flow

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

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

The channel width can be larger than the grid element and may encompass several elements (Figure 8). If the channel width is greater than the grid element width, the model extends the channel into neighboring grid elements. A channel may be 1000 ft (300 m) wide and the grid element only 300 ft (100 m) square. The model also makes sure that there is sufficient floodplain surface area after extension. The channel interacts with the right and left bank floodplain elements to share discharge. Each bank can have a unique elevation. If the two bank elevations are different in the CHAN.DAT file, the model automatically splits the channel into two elements even if the channel would fit into one grid element.

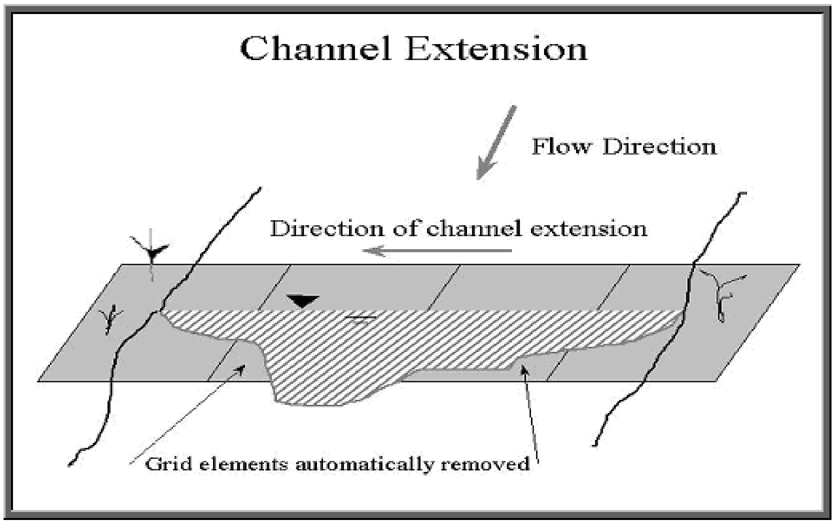


Figure 8. Channel Extension over Several Grid Elements

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

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

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

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

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

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

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

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

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

4.4 Channel-Floodplain Interface

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

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

4.5 Limiting Froude Numbers

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

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

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

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

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

percent change from the original n-value	n-value increment increase
< 20	0.0002
20 < % < 50	0.0001
50 < % < 100	0.00002
100 < % < 200	0.000002

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

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

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

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

4.6 Levees

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

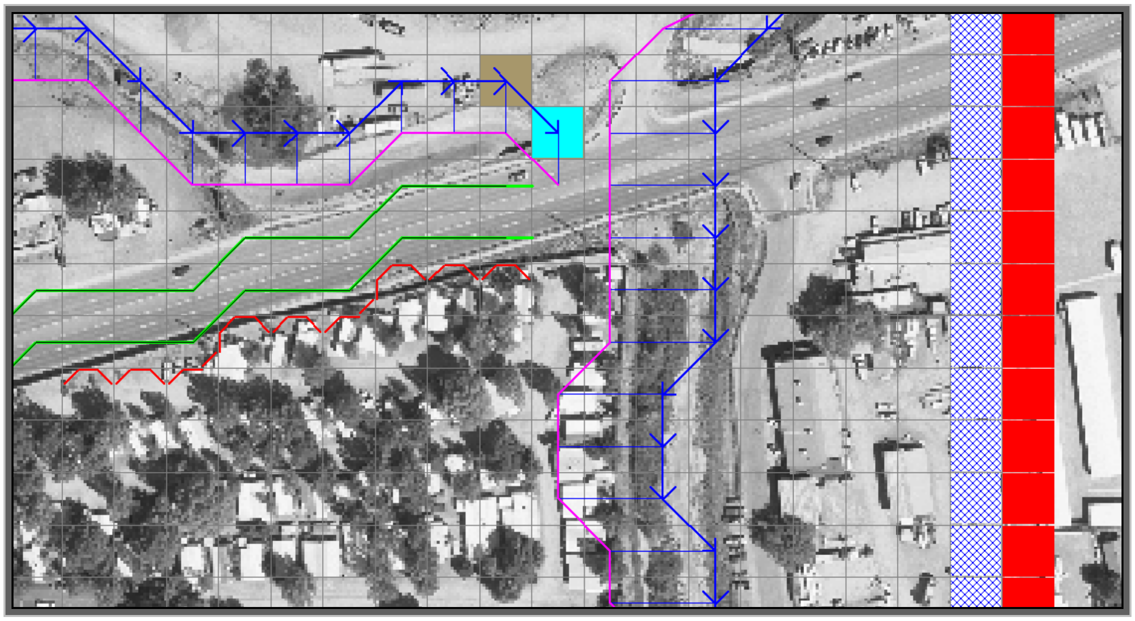


Figure 9. Levees are depicted in Red and the River in Blue in the GDS Program

The levee output files include LEVOVERTOP.OUT and LEVEEDEFIC.OUT. A discharge hydrograph overtopping the levee element is reported in LEVOVERTOP.OUT. The discharge is combined for all the levee directions that are being overtopped. Finally the LEVEEDIFIC.OUT file lists the levee elements with loss of freeboard during the flood event. Five levels of freeboard deficit are reported:

0 = freeboard > 3 ft (0.9 m)
1 = 2 ft (0.6 m) < freeboard < 3 ft (0.9 m)
2 = 1 ft (0.3 m) < freeboard < 2 ft (0.6 m)
3 = freeboard < 1 ft (0.3 m)
4 = levee is overtopped by flow.

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

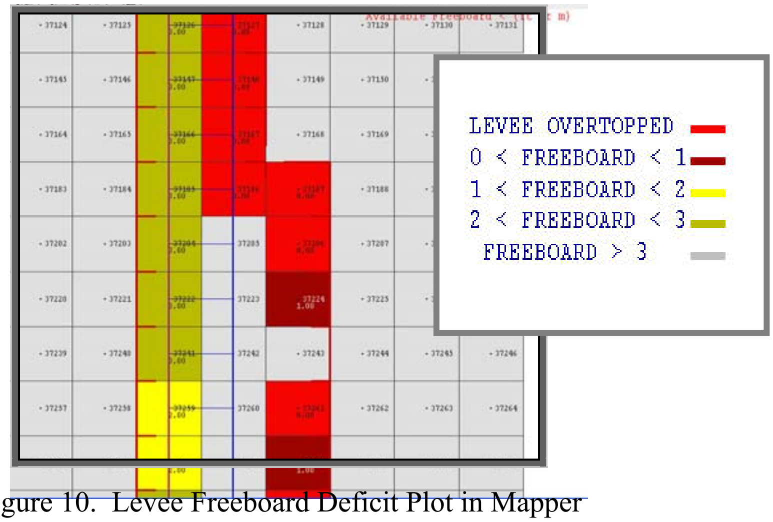


Figure 10. Levee Freeboard Deficit Plot in Mapper

4.7 Levee and Dam Breach Failures

The FLO-2D Pro model can simulate levee and dam breach failures (Figures 11 and 12), but this feature is not available in the Basic model. In the Pro model, there are two failure modes; one is a simple uniform rate of breach expansion and the other predicts the breach erosion. For both cases, the breach timestep is controlled by the flood routing model. The breach discharge is computed and the change in upstream storage volume is conserved. The breach simulation also considers the tailwater and backwater effects, and predicts the downstream flood routing. The model reports of the time of breach or overtopping, the breach hydrograph, peak discharge through the breach, and breach parameters as a function of time. Additional output files that define the breach hazard include the time to the maximum flow depth, the time to one foot flow depth and time to two foot flow depth which are useful for delineating evacuation routes. For a complete discussion of the dam and levee breach component refer to the Pro model reference manual.

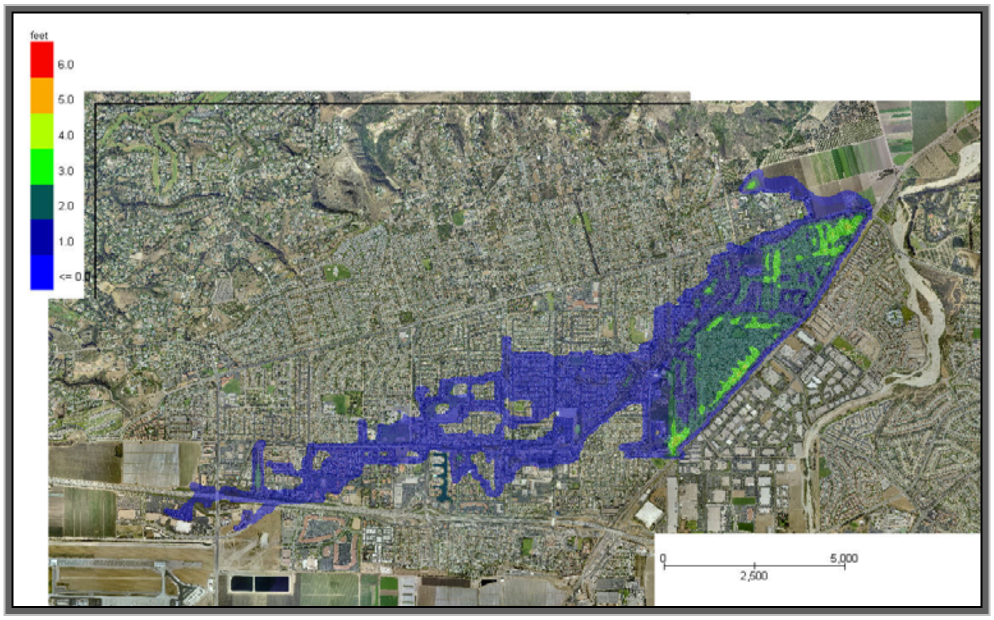


Figure 11. Example of Levee Breach Urban Flooding

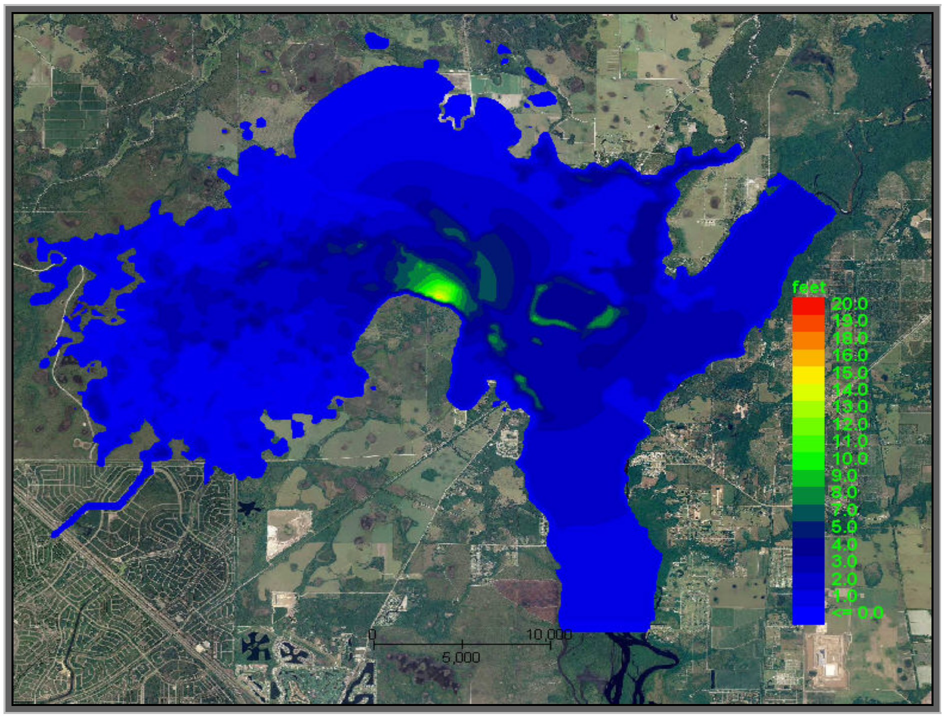


Figure 12. Example of a Proposed Domestic Water Supply Reservoir Breach Failure

4.8 Hydraulic Structures

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

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

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

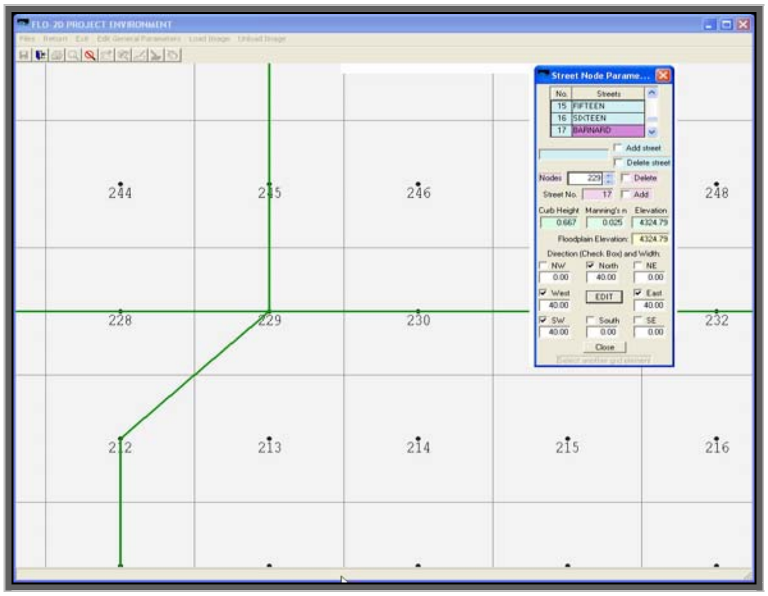
where: (a) is a regression coefficient and (b) is a regression exponent. More than one power regression relationship may be used for a hydraulic structure by specifying the maximum depth for which the relationship is valid. For example, one depth relationship can represent culvert inlet control and a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

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

4.9 Street Flow

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

Streets are assumed to emanate from the center of the grid element to the element boundary in the eight flow directions (Figure 13). For example, an east-west street across a grid element would be assigned two street sections. Each section has a length of one-half the grid element side or diagonal. A given grid element may contain one or more streets and the streets may intersect. Street roughness values, street widths, elevations and curb heights can be modified on a grid element or street section basis in the GDS program.



No.	Streets
15	FIFTEEN
16	SIXTEEN
17	BARNARD

☐ Add street
☐ Delete street
Nodes 229 ☐ Delete
Street No. 17 ☐ Add

Curb Height	Manning's n	Elevation
0.667	0.025	4324.79

Floodplain Elevation: 4324.79

Direction (Check Box) and Width:

☐ NW 0.00	☑ North 40.00	☐ NE 0.00
☑ West 40.00		☑ East 40.00
☑ SW 40.00	☐ South 0.00	☐ SE 0.00

Figure 13. Streets Depicted in Green in the FLOENVIR Program.

4.10 Floodplain Surface Storage Area Modification and Flow Obstruction

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

These factors can greatly enhance the detail of the flood simulation through an urban area. Area reduction factors are specified as a percentage of the total grid element surface area (less than or equal to 1.0). Width reduction factors are specified as a percentage of the grid element side (less than or equal to 1.0). For example, a wall might obstruct 40% of the flow width of a grid element side and a building could cover 75% of the same grid element (Figure 14).

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

NODE 103

☑ Completely blocked grid (no flow)

ARF - Area Reduction Factor (70% Surface area loss: ARF = 0.7)

ARF 1.00

WRF - Width Reduction Factor (only 4 directions per grid element)

WRF North 1.00

WRF NE 1.00

WRF East 1.00

WRF SE 1.00

Figure 14. Area and Width Reduction Factors

4.11 Rainfall and Runoff

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

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

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

Historical storms can be assigned on a grid element basis using real rainfall data. If calibrated Next-Generation Radar (NEXRAD) data is available, the rainfall on the NEXRAD pixels for a given time interval can be automatically interpolated to the FLO-2D grid system using the GDS. Each grid element will be assigned a rainfall total for the NEXRAD time interval and the rainfall is then interpolated by the model for each computational timestep. The result is spatially and temporally variable rainfall-runoff from the grid system. As example of the application of NEXRAD rainfall on an alluvial fan and watershed near Tucson. Arizona is shown in Figure 15. You can accomplish the same result with gridded network data from a system of rain gages. After the GDS interpolation, each FLO-2D grid element will have a rainfall hyetograph to represent the storm. This is the ultimate temporal and spatial discretization of a storm event and the flood replication has proven to be very accurate.

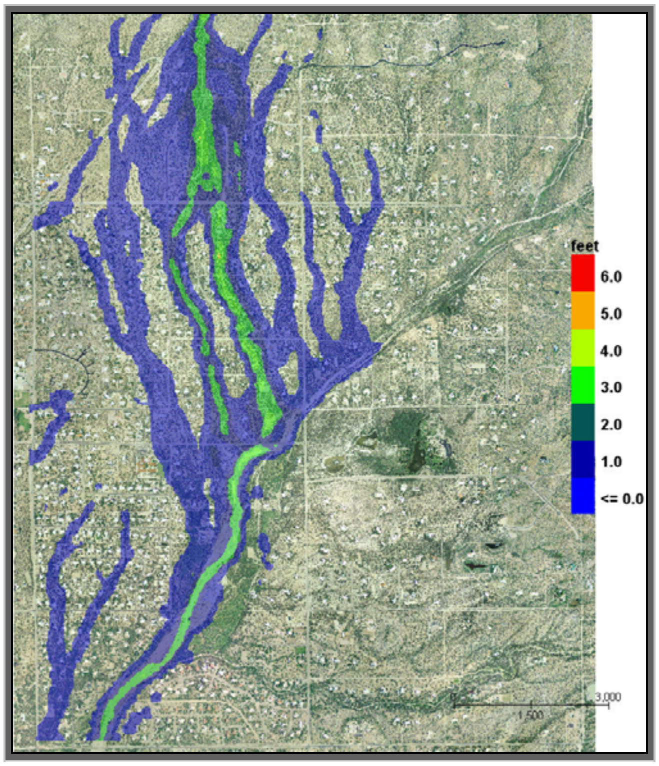


Figure 15. Flooding Replicated from NEXRAD Data near Tucson, Arizona

4.12 Infiltration and Abstraction

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

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

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

Table 2. Initial Abstraction

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

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

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

Table 3. Green Ampt Infiltration - Hydraulic Conductivity and Porosity

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

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

Table 4. Green Ampt Infiltration - Soil Suction

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

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

Table 5. Green Ampt Infiltration -Volumetric Moisture Deficiency

Classification	Dry (% Diff)	Normal (% Diff)
sand and loamy sand[1]	35	30
sandy loam	35	25
loam	35	25
silty loam	40	25
silt	35	15
sandy clay loam	25	15
clay loam	25	15
silty clay loam	30	15
sandy clay	20	10
silty clay	20	10
clay	15	5

[1] Maricopa County Drainage Design Manual, 1992.

The SCS runoff curve number (CN) loss method is a function of the total rainfall depth and the empirical curve number parameter which ranges from 1 to 100. The rainfall loss is a function of hydrologic soil type, land use and treatment, surface condition and antecedent moisture condition. The method was developed on 24 hour hydrograph data on mild slope eastern rural watersheds in the United States. Runoff curve numbers have been calibrated or estimated for a wide range of urban areas, agricultural lands and semi-arid range lands. The SCS CN method does not account for variation in rainfall intensity. The method was developed for predicting rainfall runoff from ungaged watersheds and its attractiveness lies in its simplicity. For large basins (especially semi-arid basins) which have unique or variable infiltration characteristics such as channels, the method tends to over-predict runoff (Ponce, 1989).

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

4.13 Evaporation

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

4.14 Overland Multiple Channel Flow

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

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

4.15 Sediment Transport – Total Load

Sediment transport is not available in the Basic Model. The Pro Model simulates sediment transport. As a brief overview, when a channel rigid bed analysis is performed, any potential cross section changes associated with sediment transport are assumed to have a negligible effect on the predicted water surface. The volume of storage in the channel associated with scour or deposition is relatively small compared to the entire flood volume. This is a reasonable assumption for large river floods on the order of a 100-year flood. For large rivers, the change is flow area associated with scour or deposition will negligible effect on the water surface elevation for flows exceeding the bankfull discharge. On steep alluvial fans, several feet of scour or deposition will usually have a minimal effect on the flow paths of large flood events. For small flood events, the potential effects of channel incision, avulsion, blockage, bank or levee failure and sediment deposition on the flow path should be considered.

To address mobile bed issues, the FLO-2D Pro Model has a sediment transport component that can compute sediment scour or deposition. Within a grid element, sediment transport capacity is computed for either channel, street or overland flow based on the flow hydraulics. The sediment transport capacity is then compared with the sediment supply and the resulting sediment excess or deficit is uniformly distributed over the grid element potential flow surface using the bed porosity based on the dry weight of sediment. For surveyed channel cross sections, a non-uniform sediment distribution relationship is used. There are eleven sediment transport capacity equations that can be applied in the Pro. Sediment routing by size fraction and armoring are also options. Sediment continuity is tracked on a grid element basis. For a complete discussion of sediment transport, please refer to the Pro Model reference manual.

4.16 Mud and Debris Flow Simulation

Simulating mud and debris flows is an optional component in the FLO Pro model. This component is not available in the Basic Model. As a brief overview, very viscous, hyperconcentrated sediment flows are generally referred to as either mud or debris flows. Mudflows are non-homogeneous, 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.

Hyperconcentrated sediment flows range from water flooding to mud floods, mudflows and landslides. The distinction between these flood events depends on sediment concentration measured either by weight or volume (Figure 16). Sediment concentration by volume expressed as a percentage is the most commonly used measure. Table 6 lists the four different categories of hyperconcentrated sediment flows and their dominant flow characteristics. For a complete discussion on hyperconcentrated sediment flows, refer to the Pro Model reference manual or the document *"Simulating Mudflows_Guidelines.doc"*.

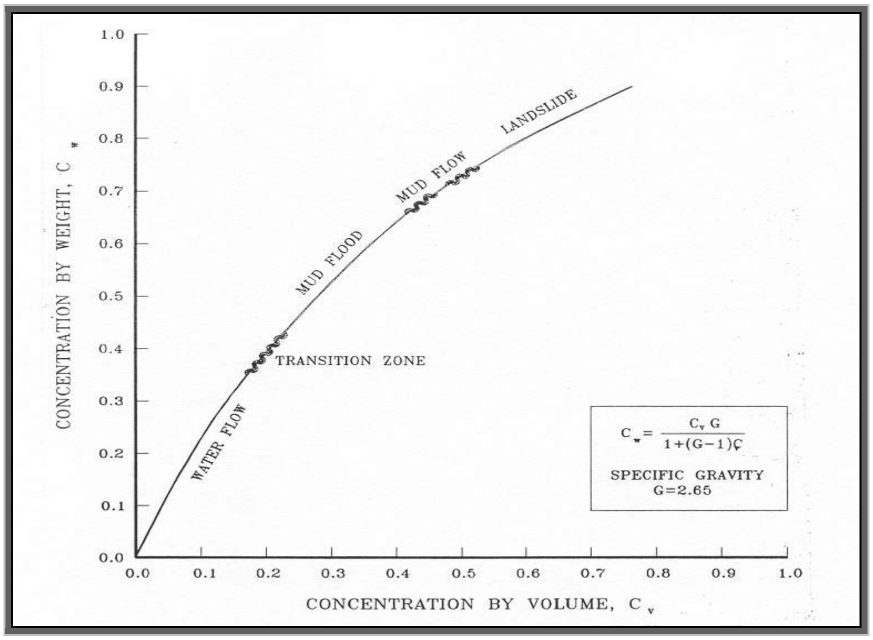


Figure 16. Classification of Hyperconcentrated Sediment Flows

Table 6. Mudflow Behavior as a Function of Sediment Concentration

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

V. FLO-2D APPLICATIONS AND METHODS

5.1 River Applications

Simulating river flow is a common application of the FLO-2D model (Figure 17). Abrupt cross section transitions, flat bed slopes, confluences and limited data bases are issues related to channel flow. The key to simulating river flooding is correctly assessing the relationship between the flood volume in the channel and the volume distributed on the floodplain. There are several considerations to defining channel volume and geometry. The surveyed channel cross sections should be appropriately spaced to model transitions between wide and narrow cross sections. The estimate of the total channel length (sum of the channel grid element lengths) is important to channel volume computation. Finally, surveyed water surface elevations at known discharges are needed to calibrate the channel roughness values. Channel routing with poorly matched channel geometry and estimated roughness can result in discharge surging.

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

The two most important FLO-2D results are the channel hydrograph at a downstream location and the floodplain area of inundation. Typically if the area of inundation is correct, then the channel flow depths and water surface elevations will be relatively accurate. Replicating the channel hydrograph and the floodplain inundation while conserving volume is a good indication that the volume distribution between the channel and the floodplain is reasonable. Channel flow routing in the Basic Model includes simulating hydraulic structures, levees, infiltration. Hydraulic structures may include bridges, culverts, weirs, diversions or any other channel hydraulic control. Levees are usually setback from the river on the floodplain. Channel infiltration is based solely on the hydraulic conductivity and represents average bed and bank seepage conditions. Channel mobile bed and mudflows are available only in the Pro Model.

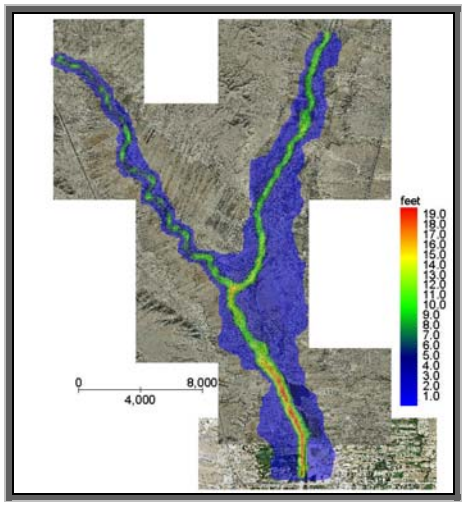


Figure 17. Middle Rio Grande and Rio Chama Confluence Model

5.2 Unconfined Overland and Alluvial Fan Flooding

The primary focus of an unconfined flood simulation is how the volume is distributed over the floodplain surface. The flood volume controls the area of inundation (Figure 18). Important flood routing details include topography, spatial variation in infiltration and roughness, flow obstructions, levees, hydraulic structures and streets. The floodwave progression over the floodplain can be adjusted with the floodplain n-values. Street flow may control shallow flooding distribution in urban areas. Buildings and walls that obstruct flow paths and or eliminate floodplain storage (Figure 19). The levee routine can be used to simulate berms, elevated road fill, railroad embankments or other topographic features to confine the flow on the floodplain. Hydraulic conveyance facilities such as culverts, rainfall and gully flow may control the local water surface elevations.

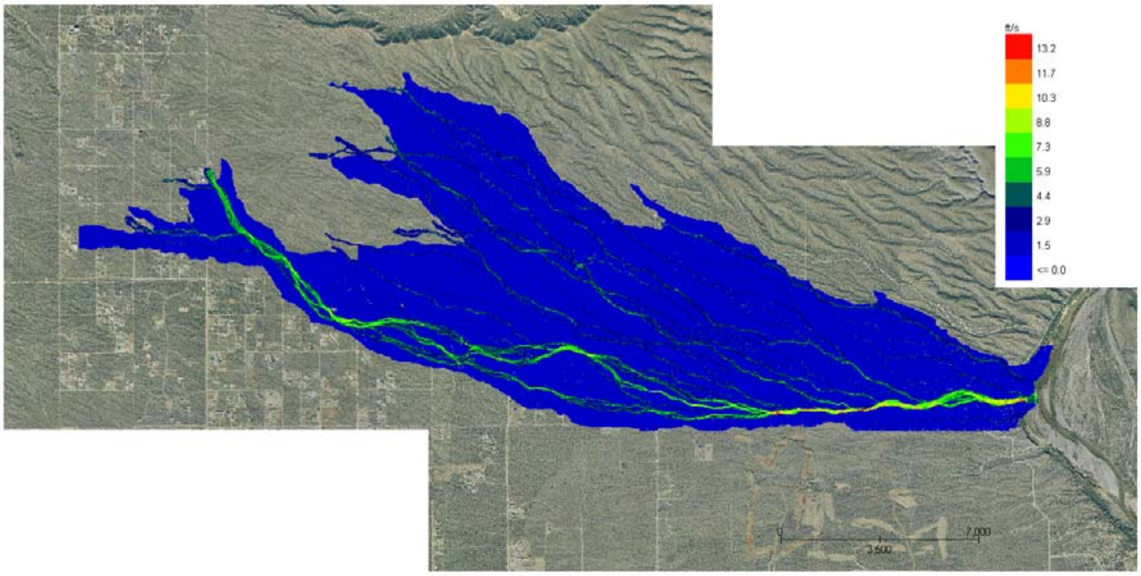


Figure 18. Unconfined Alluvial Fan Flooding

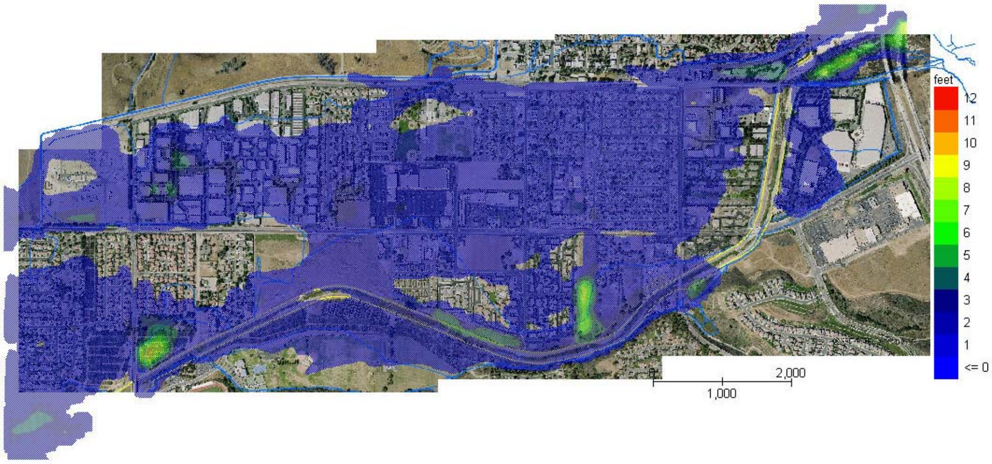


Figure 19. Urban flooding with Street Flow and Building Obstruction

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

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

5.3 Model Results – What Constitutes a Successful Flood Simulation?

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

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

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

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

VI. FLO-2D MODEL VALIDATION

The FLO-2D model has been applied on numerous projects by engineers and floodplain managers worldwide. Many users have performed validity tests including physical model studies. In January 1999, the Sacramento District Corps of Engineers conducted a review of several model applications and submitted a certification letter to FEMA. In early 2001, the Albuquerque District of the Corps of Engineers also completed a review of the FLO-2D model for riverine studies and submitted an acceptance letters to FEMA in support of using the FLO-2D for flood insurance studies. FLO-2D has been on FEMA's list of approved hydraulic models since 1999. A verification document was prepared for the FEMA submittal for the Basic Model and is available at the FLO-2D website (www.flo-2d.com).

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

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

- Channel flow in the mild sloped California Aqueduct;
- Channel flow results compared to HEC-2 model results;
- Channel and floodplain flow routing for an actual river flood, Truckee River;
- Channel routing in a large river system (Green River) with a dam release floodwave;
- Comparison of floodplain inundation with mapped wetted acreage (Middle Rio Grande);
- Verification of mudflow hydraulics through replication of a know event (Rudd Creek);
- Flume discharge for steady, uniform flow using the overland flood routing component (compared with the analog results);
- Channel replication of measured river gaging discharge (Rio Grande, Figure 20).

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

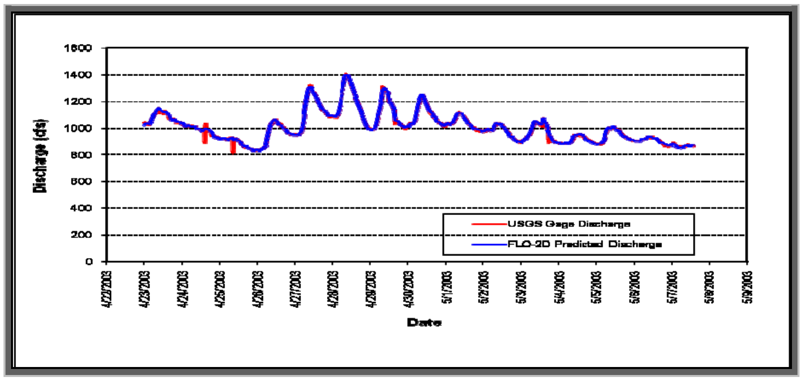


Figure 20. FLO-2D versus USGS Measured Gage Data

VII. REFERENCES

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

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.

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

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.

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.

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

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

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

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

Jin, M. and D.L. Fread, 1997. “Dynamic flood routing with explicit and implicit numerical solution

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

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