Numerical modeling study on the potential impacts of hydraulic structures in the Guayas watershed, Ecuador

ZHU Zhenduo
Ven Te Chow Hydrosystems Laboratory
Department of Civil and Environmental Engineering
University of Illinois at Urbana-Champaign, Illinois, US

Viviana MORALES
Ven Te Chow Hydrosystems Laboratory
Department of Civil and Environmental Engineering
University of Illinois at Urbana-Champaign, Illinois, US

Talia E. TOKYAY
Ven Te Chow Hydrosystems Laboratory
Department of Civil and Environmental Engineering
University of Illinois at Urbana-Champaign, Illinois, US

Marcelo H. GARCIA
Ven Te Chow Hydrosystems Laboratory
Department of Civil and Environmental Engineering
University of Illinois at Urbana-Champaign, Illinois, US

ABSTRACT: The Guayas watershed is a key region for economic development in Ecuador because of abundant aquatic product in this area. However, due to seasonality of the rain distribution, its production capacity has been badly affected by the problems related with flooding, sediment transport and inefficient natural drainage. In order to overcome those problems, a series of hydraulic projects are planned to be built in the southeast of the Guayas watershed including three sub-basins, the Bulubulu River, the Cañar River and the Naranjal River with a total extension of around 5,000 km². This study focuses on one of those projects, a diversion dam located in the Cañar River. This hydraulic structure aims at diverting a portion of the flow of Cañar River from its natural course so that the flood control ability in downstream region can be improved. In this study, the potential impact of hydraulic structure on the flow regime and bed-morphodynamics will be analyzed based on results from 1D, 2D and 3D numerical models. The results will be reported in terms of changes in the water surface elevation, velocity field, and bed-shear stress.

1 INTRODUCTION

This study focuses on Cañar River and its three tributaries, Patul, Piedras and Norcay Rivers in Ecuador. The domain of interest is approximately 38 km long. In Figure 1, the change in the bed elevation in the full domain is given. Upstream of the domain is situated in a mountainous area therefore the bed elevation changes quite dramatically over the first 10-15km of the domain. The maximum expected flow discharge in Cañar River is estimated to be around 1700 m³/s. Due to variation of rainfall distribution over the year, the region experiences flooding events frequently. The natural drainage is poor in the watershed and furthermore, Cañar River is observed to have high sediment transport capacity. The local authorities have
carried out geological and geotechnical surveys and designed a diversion dam to be built in the area, which aims to reduce the discharges in Cañar River in high flow seasons. The anticipated location of the planned structure is shown in Figure 1. Even though a detailed topographical survey of the area is provided, unfortunately field data in terms of water surface elevation, flow discharges and velocities are found lacking.

With this limitation, the objectives of our study is to understand hydraulic response of the Cañar River to possible flow conditions by simulating several scenarios using numerical models and to assess the performance of 1D, 2D and 3D hydrodynamic models in terms of feasibility and in particular their ability to predict inundation extent and flow features.

The models used reflect a move in recent years from 1D approach (represented by the US Army Corps of Engineers HEC-RAS model) to 2D and 3D modeling of open channel and river flows.

![Figure 1: Extent of simulation domain for various numerical models used in the study and evolution of bed elevation in Cañar River and its tributaries.](image)

2 METHODOLOGIES

Different hydrodynamics and morphodynamics models including HEC-RAS, HydroSed2D, EFDC and FLUENT are applied and compared herein in order to get satisfactory results under the condition that the availability of field data is limited for model calibration.

2.1 1D HEC-RAS model

HEC-RAS (River Analysis System) is a standard riverine 1D numerical model that solves the full 1D St. Venant equations for unsteady open channel flow (HEC-RAS User’s Manual, 2010). This model was developed by the United States Army Corps of Engineers and is considered to be one of the most widely used floodplain hydraulics models in the world (Wurbs and James, 2002). It has been applied extensively in flooding control projects (Horritt, 2002) and to analyze the impact of hydraulic structures in the river hydraulics (Maingi and Marsh, 2001). HEC-RAS enables one-dimensional steady and unsteady flow water surface profiles calculations; sediment transport computations; and water quality analysis. Capabilities that allow the engineers to simulate the hydraulics of the floodplain, evaluate existing conditions, determine proper design of hydraulic structures, and assess the effects of the structures. The hydraulic model requires the flow discharge as input; its parameters are representative cross-sections for each sub-basin, including left and right bank locations, roughness coefficients (Manning’s n), and contraction and expansion coefficients.

2.2 Two-dimensional in-house research code HydroSed2D

The HydroSed2D model was developed in the Hydrosystems Lab at UIUC (Liu et al., 2008). The code is open source to public. The model solves shallow water equations by using finite volume method based
on Godunov scheme. It also has the capability to solve bed-load transport equation and morphodynamic equation. Roe’s approximate Riemann solver (Roe, 1981; Roe, 1986) was applied to compute the inviscid fluxes and the use of unstructured meshes allows the applicability to irregular-boundary domains. In order to achieve higher-order accuracy, the MTNMOD TVD limiter (Hirsch, 1990) and the exact r formulation (Darwish and Moukalled, 2003) were used in the model. For sediment bed load transport, either Grass formula (Grass, 1981) or Meyer-Peter & Muller formula can be used. Model could also calculate bed evolution using Exner equation. This model does not include turbulence calculations for the flow.

The non-linear shallow water equations are expressed as follow.

\[
\frac{\partial (hu)}{\partial t} + \frac{\partial (huu)}{\partial x} + \frac{\partial (hvv)}{\partial y} = 0 \tag{1}
\]

\[
\frac{\partial (hu)}{\partial t} + \frac{\partial (huu)}{\partial x} + \frac{\partial (hvu)}{\partial y} - \nu \left( \frac{\partial (h \partial u)}{\partial x} + \frac{\partial (h \partial v)}{\partial y} \right) = \tau_{ux} - \tau_{uy} - gh \frac{\partial \xi}{\partial x} + hfy
\]

\[
\frac{\partial (hv)}{\partial t} + \frac{\partial (hvu)}{\partial x} + \frac{\partial (hvv)}{\partial y} - \nu \left( \frac{\partial (h \partial u)}{\partial x} + \frac{\partial (h \partial v)}{\partial y} \right) = \tau_{wx} - \tau_{wy} - gh \frac{\partial \xi}{\partial y} - hfu
\]

where \( \xi \) is the free surface elevation; \( h \) is water depth; \( u \) and \( v \) are depth-averaged velocities in the \( x \) and \( y \) directions, respectively; \( \tau_{ux} \) and \( \tau_{wy} \) are bed shear stresses caused by wind; \( \tau_{ox} \) and \( \tau_{oy} \) are bed shear stresses; \( \nu \) is the water viscosity; \( g \) is gravity constant; \( f \) is the Coriolis parameter.

### 2.3 Three-dimensional EFDC model

The EFDC (Environmental Fluid Dynamics Code) model was developed at the Virginia Institute of Marine Science (Hamrick, 1992), which is a public domain code maintained by Tetra Tech, Inc. with the supported of US EPA. It has the modules of hydrodynamics model, eutrophication model and sediment contaminant transport model. The model has been widely used over the world in environmental fluid studies (Liu and Garcia, 2008; Ji et al., 2007; Jiang and Shen, 2009) and river morphodynamics studies (Kong, Jiang et al., 2009). One of the recent studies focuses on hydrodynamic and water quality modeling of a combined sewer overflow event observed in Chicago River (Sinha et al. 2010). The model uses a curvilinear-orthogonal horizontal grid and a sigma vertical grid. In hydrodynamics part, it solves the three-dimensional free surface flow equations for continuity, momentum, salinity and temperature. The governing equations are as follow.

\[
\frac{\partial (mHu)}{\partial t} + \frac{\partial (mHuu)}{\partial x} + \frac{\partial (mHuv)}{\partial y} + \frac{\partial (mHvv)}{\partial z} = \left( mf + v \frac{\partial (m \partial u)}{\partial x} - u \frac{\partial (m \partial u)}{\partial y} \right) Hv \tag{2}
\]

\[
\frac{\partial (mHv)}{\partial t} + \frac{\partial (mHvu)}{\partial x} + \frac{\partial (mHvv)}{\partial y} + \frac{\partial (mHvv)}{\partial z} = \left( mf + v \frac{\partial (m \partial v)}{\partial x} - u \frac{\partial (m \partial v)}{\partial y} \right) Hv \tag{3}
\]

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \tag{4}
\]

\[
\frac{\partial (m_w)}{\partial t} + \frac{\partial (m_w u)}{\partial x} + \frac{\partial (m_w v)}{\partial y} = 0 \tag{5}
\]

\[
\frac{\partial (m_z)}{\partial t} + \frac{\partial (m_z u)}{\partial x} + \frac{\partial (m_z v)}{\partial y} = 0 \tag{6}
\]

\[
\rho = \rho(S,T) \tag{7}
\]
\[
\frac{\partial (mHS)}{\partial t} + \frac{\partial (m_H uS)}{\partial x} + \frac{\partial (m_H vS)}{\partial y} + \frac{\partial (m_w S)}{\partial z} = \frac{\partial}{\partial z} \left( mH^{-1} A_e \frac{\partial S}{\partial z} \right) + Q_s
\]

(8)

\[
\frac{\partial (mHT)}{\partial t} + \frac{\partial (m_H uT)}{\partial x} + \frac{\partial (m_H vT)}{\partial y} + \frac{\partial (m_w T)}{\partial z} = \frac{\partial}{\partial z} \left( mH^{-1} A_e \frac{\partial T}{\partial z} \right) + Q_t
\]

(9)

where \( m_s \) and \( m_a \) are the square roots of the diagonal components of the metric tensor; \( m = m_s m_a \); \( u \) and \( v \) are the horizontal velocities in the curvilinear orthogonal coordinates \( x \) and \( y \); \( w \) is the dimensionless velocity in coordinate \( z \); \( H \) is total water depth which equals the sum of \( h \) (depth below the undisturbed free surface) and \( \zeta \) (surface elevation); \( p \) is the amount of physical pressure in excess of the reference density hydrostatic pressure normalized by the reference density \( \rho_0 \); \( f \) is the Coriolis parameter; \( A_s \) is the vertical turbulent viscosity; \( Q_s \) and \( Q_t \) are momentum source and sink terms modeled by subgrid-scale horizontal diffusion; \( S \) and \( T \) represents salinity and temperature respectively; \( Q_s \) and \( Q_t \) are the source and sink terms for salinity and temperature respectively; \( A_{vH} \) is vertical turbulent diffusivity. \( A_s \) and \( A_t \) are computed by second moment turbulent model developed by Mellor and Yamada (1982) and modified by Galperin et al. (1988).

2.4 FLUENT

FLUENT is a non-hydrostatic finite-volume commercial code that solves three dimensional Navier-Stokes Equations. It has recently been used in several studies that involve CFD modeling of three-dimensional open channel flows. Nicholas and McLelland (1999) studied the patterns of mean and turbulent flow using a combination of field measurements and numerical simulations in River Culm in Devon, UK. Ma et al. (2002) simulated flow in an upland urban river which is prone to flooding. Similarly, Dargahi (2004) investigated the flow features in River Klaralven with focus on river bifurcations using computational fluid dynamics (CFD).

In this study, free-surface flow in a short reach (see Figure 1) of Cañar River is simulated using Volume of Fluid Method (VOF). VOF method can be used to model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. Formulation relies on the fact that two or more fluids are not interpenetrating. For each additional phase added to the model, a variable is introduced, which is the volume fraction of the phase in the computational cell (ANSYS FLUENT 12.0 User Manual 2009). The interface estimation is done by Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM). CICSAM (Ubbink, 1997) is a high resolution differencing scheme. This scheme is particularly suitable for flows with high ratios of viscosities between the phases. Steady state simulation is performed using k-ε model with standard wall functions as the turbulence closure. All operators are discretized using second order upwind scheme. Gravitational affects are also taken into account in the simulation.

3 RESULTS AND DISCUSSIONS

3.1 Model Settings and Computational Grids

3.1.1 HEC-RAS model

1D HEC-RAS model was developed to simulate the whole waterways hydrodynamics. Cross-section bathymetry data for this study was obtained from the field surveying carried out by ACSAM Consulting Firm in Ecuador. A total number of 673 cross sections represented by 73,885 points were used for the analysis. Roughness coefficients were estimated with the Stickler formula based on the mean diameter of the particles in the river (Subramanya, 1982). Manning’s n of between 0.025 and 0.034 were used for the various reaches of the river, while a constant value of 0.05 was set for the floodplain. In all analyses, flow was considered to be sub-critical thus only boundary conditions for the downstream end of the river were needed. For the main river, Cañar River, an energy slope of 0.001 was used for calculating normal depth (by using Manning’s equation) at the downstream. The time step of computation is 10 seconds.

3.1.2 HydroSed2D model
In order to see the impact of hydraulic structure more clearly, a shorter reach of the river downstream of the planned location of hydraulic structure is selected as the simulation domain for HydroSed2D applications. As shown in Figure 2, triangular elements are used in computational grid. Two cases are modeled: one is using the original wall boundaries from field survey; the other one is using designed levees as wall boundaries. In the former one, total of 2442 elements are used in this 2-km-long reach, while in the latter one 1874 elements are used. Constant discharge flow conditions are simulated and the time step of computations is 0.1 second.

![Figure 2 Computational domain and unstructured meshes in HydroSed2D model (left: original boundaries; right: designed levees as boundaries)](image)

3.1.3 EFDC model

In EFDC simulations, 6985 horizontal cells and 8 vertical layers are used herein (Figure 3). The study domain is the whole Cañar River system shown in Figure 1. EFDC uses curvilinear-orthogonal meshes in horizontal directions (see Figure 3). Due to this orthogonal requirement, fitting computational mesh smoothly to real boundaries is quite challenging, while the unstructured meshing techniques used in HydroSed2D offers an advantage in generating grids for complex topographies compared to structured ones. The time step is 0.05 second.

![Figure 3 Computational domain and grid used in EFDC simulations; a) general view of Cañar River with its tributaries (black arrows shows the direction of the flow), b) eight vertical layers used in 3D EFDC simulations, c) 2D planar view of the grid near the confluence of Patul and Cañar Rivers.](image)

3.1.4 FLUENT

The computational grid for FLUENT simulation has around 500,000 hexahedral elements. The domain is approximately 17.5 km long (see Figure 4). Over the vertical direction 8 computational nodes are used. The average size of the computational cells in longitudinal and transverse directions are around $\Delta x = 13.5m$ and $\Delta y = 7.5m$, respectively. The steady state simulation has been carried out until the differences between values of variables in continuity, momentum and turbulence closure equations at iteration i and i-1 become less than the limit, which is taken as $10^{-6}$. 
3.2 Comparison between HEC-RAS, EFDC and FLUENT simulations

In the absence of field data, the performance of each model is evaluated by comparing the results of one model to another. Figure 5 shows the comparison of water surface elevations from HEC-RAS simulation to EFDC simulation, where good agreement between models is observed. The water surface elevations are compared at 5 locations marked as L1-L5 in Cañar River. In both models constant discharges of 1039.6, 528.5, 116.3 and 450 m$^3$/s are introduced at the inflow boundaries of Cañar, Patul, Piedras and Norcay Rivers, respectively. These discharges correspond to the 50-year flood event based on the design hydrographs that were constructed for each river of the watershed. The hydrographs were provided by local authorities in Ecuador and constructed based on 23 years of mean daily flow data and 24 hour maximum precipitation data for the 1964-1988 period.

![Figure 5](image_url) Water surface elevations from HEC-RAS and EFDC simulations are compared at 5 locations (L1-L5) shown in the inset.

Similar to EFDC simulation, constant inflow discharges are introduced from two inlets (Cañar and Patul) in FLUENT simulation. The near bed velocity contours from both EFDC and FLUENT simulations are given in Figure 6. The velocity values predicted by EFDC are slightly higher than the values predicted by FLUENT. Even though EFDC is a quasi-3D hydrostatic model and Fluent is a non-hydrostatic 3D model,
it is believed that the discrepancies between these simulations could be mainly due to different mesh and bathymetry resolution. The pressure distribution assumption may not be as important since the flow depths compared to width of the river is quite small. However, qualitatively the evolution of flow velocities over the streamwise direction is quite similar in both models. The bends in the river topography cause flow to deviate from one bank to the other forming low flow zones near opposite bank. An increase in velocity near left bank after downstream of confluence of Cañar and Patul Rivers is observed in both simulations. The capability of 3D models to produce such flow information near river bed and over the transverse direction is an advantage over 1D models especially in predicting potential areas of bed erosion and sediment deposition. For instance, based on simulation results, due to increase in near bed flow velocities, one can expect to observe bed erosion near left bank after the confluence of Patul and Cañañ Rivers between 25<x<30 km as shown in Figure 6, in a flooding event where total discharge from these two rivers are around 1500m³/s.

In the next section we concentrate on results from EFDC and HydroSed2D simulations to better understand the changes in flow patterns in the river for different flow conditions.

3.3 EFDC simulations

In order to better understand river hydraulics, two scenarios are simulated using EFDC model. The first case considers constant inflow discharges for Cañar River and its tributaries as given in previous section. The second case uses synthetic design hydrographs corresponding to 50-year flood given in Figure 8.

3.3.1 Constant Discharge Case

The constant discharge simulation shows the effect of a flood where total amount of discharge downstream of Norcay and Cañar confluence is approximately 2100m³/s. In such an event, the depth-averaged flow velocity distribution, the flow velocities near the bed and change of water depth in streamwise direction are shown in Figure 7.

![Figure 7](image-url)

Figure 7  Contour plots of a) depth averaged velocity magnitude; b) bottom layer velocity magnitude and c) water depth from 3D EFDC simulations with constant inflow discharges at Cañar River and its tributaries. The close-ups show the region where the diversion dam will be constructed.
Especially the information near bed could give better evaluation for sediment transport and river bed evolution. Over the steep region in upstream Cañar River Flow, between 24<x<38 km, the flow velocities are significantly higher compared to values downstream. From upstream due to high flow velocities, large amount of sediment could be transported with flood and brought to downstream Cañar River, where water depth increases significantly downstream of Norcay intersection (see Figure 7c) and flow velocity reduces. The stretch of the river where x<10km has higher water depths and lower flow velocities which suggests possible deposition of sediment carried from upstream in this region. The contour plots near proposed construction site are shown in the close-ups on right column of Figure 7. Near bed velocities are estimated to be higher near left bank and water depth is around 2m in the mid-channel and is around 1m near the banks in this region. Such information is also useful in predicting possible local erosion prone areas.

3.3.2 Design Hydrograph Case

![Figure 8](image)

Figure 8  Synthetic design flow hydrograph of Cañar River and its tributaries for a 36-hr period based on rainfall distribution data of the region. The dashed lines show the time instances that the simulation results are reported.

In Figures 9 and 10, water depth at t = 12hr, 18hr and 24hr are shown from the simulation which uses the flow hydrographs (see Figure 8) as inflow boundary condition at each river inlet. At these selected time instances, we can expect to see the beginning, peak and end of the flooding event. The close-ups on the right column of Figure 9 show the proposed construction site for the diversion dam, while the ones in Figure 10 show downstream of Cañar-Norcay confluence. Figures show how the water depth in river could change rapidly over 12-hr period in case of such an event given in Figure 8. The figures also suggest that in terms of flood-plain protection the levee heights should be minimum 2m in the upstream and they should be above 4m in the downstream of the river.

![Figure 9](image)

Figure 9  Contour plots of water depth at a) t=12hr, b) t=18hr and c) t=24hr in the simulation domain and near the construction site.  Results are from 3D EFDC simulation with inflow conditions based on design hydrographs given in Figure 8.
3.4 HydroSed2D simulation

The simulation domain for HydroSed2D applications starts downstream of planned diversion dam and reaches to the confluence where Piedras River meets Cañar River. Two simulation scenarios are modeled with constant inflow $Q = 1568.1\text{m}^3/\text{s}$ and $Q = 468.1\text{m}^3/\text{s}$. The high discharge case simulates a flood condition that combines the inflows from Cañar and Patul Rivers; while the low discharge case represents the expected effect of hydraulic structure, which is designed to divert about 1100$m^3$/s flow into a by-pass channel in case of an upstream flood with $Q \sim 1500$m$^3$/s or higher. The downstream water surface elevation conditions are taken from the results of EFDC simulations and given as boundary conditions in HydroSed2D simulations. In rest of the domain the water depth and flow velocity is evaluated based on upstream discharge and downstream water surface elevation conditions.

The results of HydroSed2D simulations in terms of depth-averaged velocity magnitude and water depth for both scenarios are shown in Figure 11. The code has the capability to predict dry areas in low flow situations. Some dry areas can be seen clearly in low discharge case especially near left bank (see Figure 11c). In low discharge case, the water depth ranges between 0.5-2m in the reach while in high flow case it mostly changes between 2-3m. Similarly, flow velocities are reduced acutely in simulation where diversion dam is assumed to divert 1100$m^3$/s of flow from this reach compared to high discharge case.

Figure 11  HydroSed2D simulation results of depth-averaged velocity magnitude and water depth for $Q = 1568.1\text{m}^3/\text{s}$ (b, d) and $468.1\text{m}^3/\text{s}$ (a, c).
Even though the dry areas can be seen in the case of low discharge in figure 11, flood areas still exist which may result in troubles. Together with a diversion dam series of levees are designed to be placed downstream of the dam in Cañar River to further prevent flooding in the region. In Figure 12, the water surface elevation contours are given together with the designed levees for low discharge case. In this simulation the locations of designed levees are taken as wall boundaries instead of the original boundaries. The elevation of the levees is shown using same color scale as water surface elevation. The effectiveness of the levees could be assessed using simulation results in this reach. If one assumes that diversion dam works as designed, simulation results show that the levees are high enough to avoid water inundation to flood plain.

Figure 12 HydroSed2D simulation result of water surface elevation with the designed levees (represented by colorful squares).

4 CONCLUSIONS

In this study, hydraulic response of Cañar River in Ecuador to several flow conditions is assessed using numerical modeling techniques. This paper reports results mostly from three-dimensional simulations using EFDC model and two-dimensional simulations using in-house research code HydroSed2D. These simulations could be considered as a primary look to Cañar River system in attempt to understand the changes in water depth, flow velocities and possible erosion/deposition sites for sediment under different flow conditions. One-dimensional simulations provided a general idea on water depth and flow velocities that one can expect for certain flood scenarios, while three-dimensional simulations provided a more detailed view on how flow could change in transverse direction and near bed regions. However, the attempt of simulating the full reach of the river using three-dimensional methods carries challenges in terms of grid generation, simulation time and choosing appropriate turbulence closure model. On the other hand two-dimensional approach provides more detailed information than one-dimensional simulations with fewer challenges than the three-dimensional ones. The integral part of river flow modeling is the accurately collected useful field data, which is unfortunately lacking in this study. The field data is very necessary in calibrating and validating any river flow simulations.

As future work, we will concentrate on the diversion dam and changes in flow conditions with structure in place using three-dimensional approach in a smaller region near proposed construction site in Cañar River. In-house research code has recently been improved with the implementation of a new suspended sediment module, therefore, new simulations will be carried out using this module to further understand the sediment transport and flow patterns in the river. Even though it is hard to illustrate the grid independency in natural river problems, finer grids will be tested for similar boundary conditions reported in this paper as part of the future work.
5 ACKNOWLEDGEMENTS

This work was financially supported by SENAGUA (National Secretary of Water in Ecuador), ACSAM Consulting Firm, and CT Consulting Firm. The HEC-RAS model for the rivers and the experimental data was provided by the research and academic unit PROMAS (Soil and Water Management Program) from the University of Cuenca-Ecuador. The field data was provided by ACSAM Consulting Firm. The support of the Chester and Hellen Siess Professorship at the University of Illinois is gratefully acknowledged.

REFERENCES

Fluent (ANSYS), 2009, Fluent 12.0 User’s Guide.
Hanrick J. M. 1992, A three-dimensional environmental fluid dynamics computer code: theoretical and computational aspects. The College of William and Mary, Virginia, College of Science and Engineering, Davis CA.