Conference Paper, Published Version

Altinakar, M. S.; McGrath, M. Z.; Ramalingam, V. P.; Omari, H.; Miglio, E.

An Enhanced Two Dimensional Numerical Model for Simulating Floods Due to Dam And Levee Break/Breaching

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Kuratorium für Forschung im Küsteningenieurwesen (KFKI)

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/109819

Vorgeschlagene Zitierweise/Suggested citation:
Altinakar, M. S.; McGrath, M. Z.; Ramalingam, V. P.; Omari, H.; Miglio, E. (2010): An Enhanced Two Dimensional Numerical Model for Simulating Floods Due to Dam And Levee Break/Breaching. In: Sundar, V.; Srinivasan, K.; Murali, K.; Sudheer, K.P. (Hg.): ICHE 2010. Proceedings of the 9th International Conference on Hydro-Science & Engineering, August 2-5, 2010, Chennai, India. Chennai: Indian Institute of Technology Madras.

Proceedings of Ninth International Conference on Hydro-Science and Engineering (ICHE 2010), IIT Madras, Chennai, India.
2 - 5 August 2010

AN ENHANCED TWO DIMENSIONAL NUMERICAL MODEL FOR SIMULATING FLOODS DUE TO DAM AND LEVEE BREAK/BREACHING

Altinakar M.S.[1], M.Z. McGrath [2], V.P. Ramalingam[3], H. Omari[4], and E. Miglio[5]

***Abstract**: Failure of dams and levees may lead to catastrophic floods that would cause loss-of-life and damage to urban and rural areas and may have serious consequences for the regional and/or national economy and security. Unfortunately, a large number of high-hazard and significant hazard dams in the USA do not yet have an emergency action plan (EAP), partly due to high-costs of studies using one-dimensional models. Two-dimensional models offer a cost-effective alternative but are not widely adopted by the engineering community.*

This paper describes a two-dimensional dam/levee breach flood model based on a first order shock capturing explicit scheme, which was enhanced with various advanced features to suit the needs of practicing engineers. Called CCHE2D-FLOOD, this numerical model allows definition of multiple water bodies and dams, each having its own breaching sequence. This feature can be used to simulate cascading dam failures. The user can define source and sink areas that can be used as inlets and outlets through which a discharge is introduced into or extracted from the regions of computational domain (without momentum input). Coupled source and sink areas can be defined to simulate controlled releases from appurtenances, such as bottom outlets (modeled as orifices) or spillways (modeled as weirs). CCHE2D-FLOOD offers a cut-cell boundary capability to simulate linear terrain features that cannot be captured by the resolution of the computational grid. The cut-cell boundary method is also used for providing coupled 1D-2D simulation capability. A graphical user interface (GUI) programmed as an extension to ArcGIS software by ESRI provides a user-friendly environment to set-up the model and to run the simulations. The results provided by the numerical model can be directly imported into a GIS-based post-processing module for evaluation of potential loss-of-life, urban and agricultural damage, and for preparing maps of danger criteria for humans, buildings, and vehicles, etc.

***Keywords**: Dam break; 2D flood model; cut-cell boundary; cascading failures; controlled release.*

---

1 Associate Director and Research Professor, National Center for Computational Hydroscience and Engineering, University of Mississippi, MS, USA, Email: altinakar@ncche.olemiss.edu
2 Graduate Student, National Center for Computational Hydroscience and Engineering, University of Mississippi, MS, USA, Email: mzmcgrat@ncche.olemiss.edu
3 Research Associate, National Center for Computational Hydroscience and Engineering, University of Mississippi, MS, USA, Email: virjay@ncche.olemiss.edu
4 Graduate Student, National Center for Computational Hydroscience and Engineering, University of Mississippi, MS, USA, Email: hhomari@ncche.olemiss.edu
5 Assistant Professor, Dept. of Mathematics "F. Brioschi", Politecnico di Milano, Italy, Email: edie.miglio@polimi.it; formerly visiting professor at the NCCHE, University of Mississippi, MS, USA

INTRODUCTION

Two-dimensional (2D) numerical models based on shock capturing conservative schemes offer significant advantages in simulating catastrophic floods due to dam/levee break/breaching: 1) digital elevation model can be directly used as a regular computational mesh; 2) floods with mixed flow regimes (subcritical, transcritical and critical) and wetting and drying are handled; 3) non-channelized flows over flat topography can be easily simulated; 4) disconnected flow domains are handled; and 5) simulation directly provides two-dimensional spatial distribution of flood depths, velocity vectors, discharge, flood arrival time and duration, which can be used in consequence analysis. However, 2D numerical models are not yet routinely used in engineering studies, partly due to the fact that they do not offer the functionalities needed by practicing engineers, such as controlled release from bottom outlets, flow through bridges and culverts, etc.

To bridge this gap, the existing two dimensional dam/levee break/breaching model CCHE2D-FLOOD, developed by the National Center for Computational Hydroscience and Engineering, The University of Mississippi, was enhanced by adding several new features needed by the engineers to carry out flood simulations compliant with engineering practices and standards.

The first part of the paper introduces the existing CCHE2D-FLOOD model, which includes a two-sided cut-cell boundary capability to represent linear terrain features and to represent a 1D channel flow coupled with the 2D model. More information about the CCHE2D-FLOOD model can be found in Altinakar et al. (2009a). The two-sided cut cell methodology is described in Miglio et al. (2008). The application of this method to represent linear terrain features and carry out coupled 1D-2D simulations are presented in Altinakar et al. (2009b, c and d).

The second part of the paper discusses the additional features recently implemented in CCHE2D-FLOOD model to custom tailor it to the needs of practicing engineers and the GIS-based GUI, which was designed to assist users in preparing the input data for the numerical model. These include: 1) definition of multiple dams with their breaching sequences and quadtree local refinement capability to better model the time evolution of the breach geometry independently from the mesh size; 2) definition of source and sink areas that can be used to model various realistic flow situations, such as controlled release from a reservoir, flow through culverts, etc. A GOS-based preprocessor allows the user to initialize water covered areas by taking into account cut-cell lines, dams, and boundaries. One can set up observation points, lines and profiles to store time variation of selected simulation results for further analysis. Different types of boundary conditions can be defined along the perimeter of the computational domain.

DESCRIPTION OF THE EXISTING CCHE2D-FLOOD MODEL

CCHE2D-FLOOD numerically solves two-dimensional unsteady shallow water equations, which are written in vector form as (Altinakar et al., 2009a):

$$\frac{\partial \mathbf{U}}{\partial t}+\frac{\partial \mathbf{F}(\mathbf{U})}{\partial x}+\frac{\partial \mathbf{G}(\mathbf{U})}{\partial y}=\mathbf{S}(\mathbf{U}) \qquad (1)$$

with $\mathbf{U}$ as the vector of conserved variables. Flux vectors $\mathbf{F}(\mathbf{U})$ and $\mathbf{G}(\mathbf{U})$ are defined as:

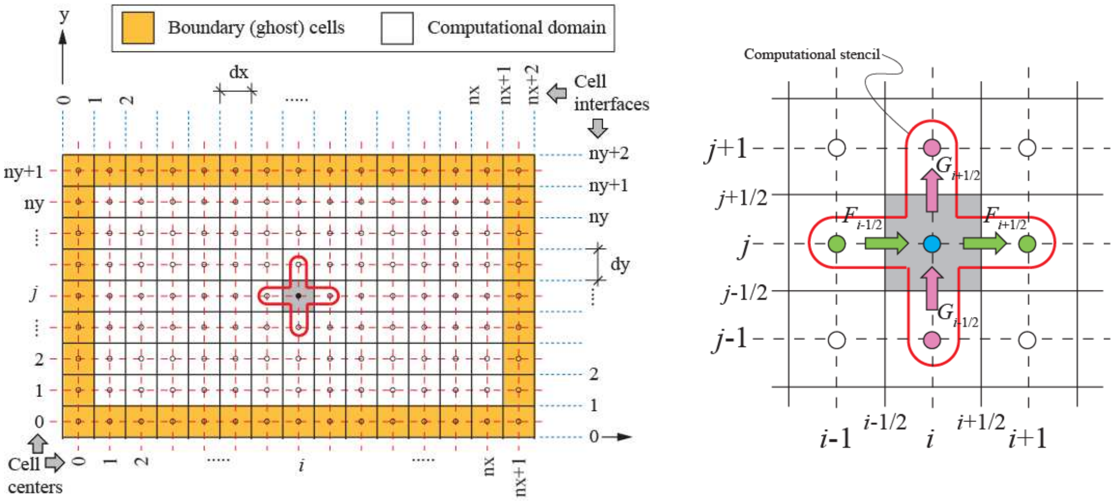


Fig. 1 Regular Cartesian computational grid and computational stencil.

$$\mathbf{U}=\begin{bmatrix} h \\ hu \\ hv \end{bmatrix}=\begin{bmatrix} h \\ Q_x \\ Q_y \end{bmatrix} \quad ; \quad \mathbf{F}=\begin{bmatrix} Q_x \\ Q_x^2/h \\ Q_xQ_y/h \end{bmatrix} \quad ; \quad \mathbf{G}=\begin{bmatrix} Q_y \\ Q_yQ_x/h \\ Q_y^2/h \end{bmatrix} \tag{2}$$

where h is water depth, u and v are the components of velocity vector in x and y directions. The volume fluxes (discharges) in x and y directions are represented by Q_x and Q_y. Including also a volume source or sink in the cell, denoted q_v, and taking Z as the water surface elevation and C the Chezy friction coefficient, the vector of source terms $\mathbf{S}(\mathbf{U})$ becomes:

$$\mathbf{S}=\begin{bmatrix} q_v \\ -gh(\partial Z/\partial x)-g\left(u\sqrt{u^2+v^2}/C^2\right) \\ -gh(\partial Z/\partial y)-g\left(v\sqrt{u^2+v^2}/C^2\right) \end{bmatrix} \tag{3}$$

Referring to the computational grid represented in Figure 1, a cell-centered finite-volume discretization of Eq. 1 over a rectangular control volume leads to the following explicit scheme:

$$U_{ij}^{n+1}=U_{ij}^{n}-\left(\Delta t/\Delta x_i\right)\left(F_{i+1/2,j}-F_{i-1/2,j}\right)-\left(\Delta t/\Delta y_i\right)\left(G_{i,j+1/2}-G_{i,j-1/2}\right)+\Delta t\; S_{ij} \tag{4}$$

where Δx and Δy represent the cell dimensions in x and y directions, and Δt is the time step. The first order upwinding method is used to compute the intercell fluxes (see Figure 1):

$$\mathbf{F}_{i+1/2j}=\begin{bmatrix} Q_x \\ Q_x^2/h \\ Q_xQ_y/h \end{bmatrix}_{i+k} \quad k=\begin{cases} 0 & Q_x\geq 0 \\ 1 & Q_x\leq 0 \end{cases}; \quad \mathbf{G}_{ij+1/2}=\begin{bmatrix} Q_y \\ Q_yQ_x/h \\ Q_y^2/h \end{bmatrix}_{j+m} \quad m=\begin{cases} 0 & Q_y\geq 0 \\ 1 & Q_y\leq 0 \end{cases} \tag{5}$$

The above upwinding relationships for intercell fluxes are supplemented by additional relationships to take into account special conditions, such as wet and dry cell interactions, converging flows, very steep ground elevations, etc. By introducing the Manning's roughness coefficient n, source terms are discretized in the following manner:

$$\mathbf{S} = \begin{bmatrix} q_v \\ -gh_{ij}^{n+1}\left[\left(\partial Z/\partial x\right)\right] - g\left(u_{ij}\sqrt{u_{ij}^{2}+v_{ij}^{2}}\,/\left(h_{ij}^{1/6}/n\right)^{2}\right) \\ -gh_{ij}^{n+1}\left[\left(\partial Z/\partial y\right)\right] - g\left(v_{ij}\sqrt{u_{ij}^{2}+v_{ij}^{2}}\,/\left(h_{ij}^{1/6}/n\right)^{2}\right) \end{bmatrix} \qquad \text{with} \qquad C = \frac{h_{ij}^{1/6}}{n} \tag{6}$$

The CCHE2D-FLOOD model handles dry-bed condition by maintaining a very small water depth, which is chosen sufficiently small not to cause any noticeable change in the propagation speed of the wet/dry front. When a wet cell and dry cell share a common edge, only the mass flux is allowed into the dry cell (momentum flux is set to zero) during that particular time step.

The CCHE2D-FLOOD is stable, robust and oscillation-free near discontinuities. It has been shown to rigorously conserve mass and momentum. No special entropy fixes are needed to ensure physically plausible solutions. CCHE2D-FLOOD has been verified and validated using analytical solutions as well as data from laboratory experiments, model tests, and past dam break events (see Ying and Wang, 2004; Ying et al., 2004, and Altinakar et al. 2010).

The CCHE2D-FLOOD uses an explicit scheme to solve the shallow water equations. It is, therefore, subjected to the Courant-Friedrichs-Lewy (CFL) condition for stability and convergence. Given a mesh size, $\Delta x \times \Delta y$, the CFL condition places an upper bound on the time step as follows:

$$N_{CFL} = \text{Max}\left[\left(\Delta t/\Delta x\right)\left(|u|+\sqrt{gh}\right),\left(\Delta t/\Delta x\right)\left(|v|+\sqrt{gh}\right)\right] \leq 1 \tag{7}$$

To preserve the positivity of flow depth, drying cells may require a smaller time step that indicated by the CHL condition. The time step is automatically selected by the program.

Cut-cell Boundary Capability to Represent Linear Terrain Features and 1D Rivers

A two-sided cut-cell boundary method is used to represent linear terrain features, such as road embankments, that may significantly affect the flood propagation but are too narrow to be adequately captured by the typical resolutions of the DEMs used in engineering studies. Using the GIS-based preprocessor, the linear features are projected (see Figure 2) onto the computational mesh following three basic rules: 1) a cell can only be cut by a single straight line; 2) The line joining the centers of two adjacent cells can only be cut by a single straight cut-line, 3) When two adjacent cells are both cut, the lines should meet at the same point on the common edge. The pre-processor checks the compliance with these rules and the user can make the changes in case of error. The projected lines cut through computational cells and divide them into two irregular-shaped parts (Figure 3). Cells with computational stencil affected by a cut line are computed using a modified form of the Ghost Fluid Method (GFM) (Ghias et al., 2007), which does not require a smaller time step to handle small irregular-shaped cell portions. When the water depth is sufficient the water may overtop a cut-cell boundary and the discharge is computed using an appropriate weir equation. Miglio et al. (2008) and Altinakar et al. (2009b) present the two-sided cut cell methodology and give examples of its use.

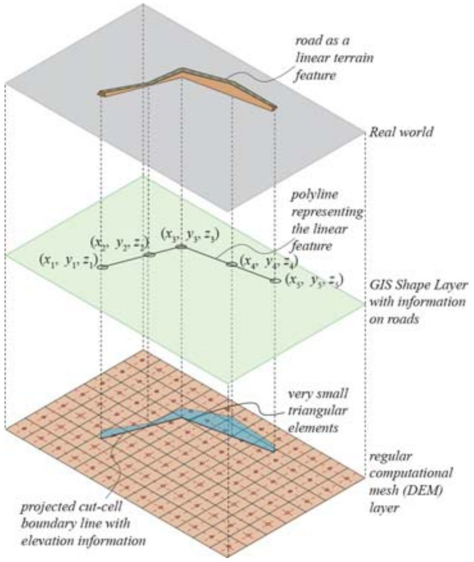


Figure 2. Projection of a linear terrain feature onto a regular Cartesian Mesh.

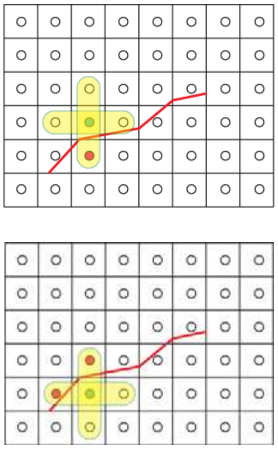


Figure 3. Computational stencils in the presence of a two-sided cut-cell boundary.

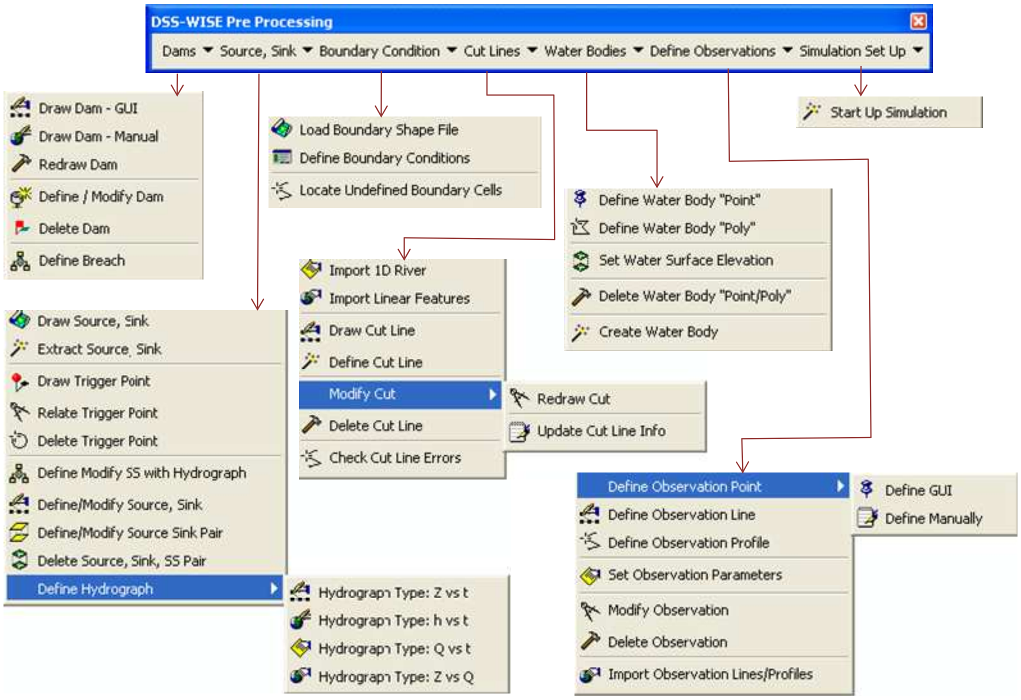


Fig. 4 Toolbar and menu structure of the preprocessor.

The cut-cell boundary method is also used to represent one-dimensional rivers, either narrow or wide. Narrow river option with a single cut line is used when the river width is less than the cell size. The model automatically distinguishes between left and right banks and calculates the discharges exchanged with the 2D model, if any. Wide river option is more appropriate when the river width is considerably larger than the cell size. In this case left and right banks of the river are represented by separate cut lines and the 2D computational cells covered by the river width are turned off. This functionality can be used to carry out coupled levee breaching simulations. The details can be found in Altinakar et al. (2009a, c and d).

CUSTOM TAILORING OF CCHE2D-FLOOD FOR ENGINEERING APPLICATIONS

The following capabilities were added to the CCHE2D-FLOOD to accommodate the requirements of engineering practice and procedures: 1) handling of multiple dams and their breaching sequences including the possibility of local mesh refinement using the quadtree method; 2) definition of sources and sinks to simulate reservoir operations and flow through structures; 3) definition of initial water bodies by taking into account the cut-lines and dams; 4) definition of observation points, lines and profiles for monitoring simulation results. These features are defined using a GIS-based preprocessor, which also allows the user to project cut lines, set the boundary conditions along the perimeter of the computational mesh, and prepare and write the final CCHE2D-FLOOD input file. Figure 4 shows the preprocessor toolbar implemented in ArcGIS and the menu structure.

Definition of Dams and Breach Sequences

If its bathymetry is known, a reservoir can be included in the 2D computational mesh. When the dam is breached the model calculates the flow through the breach, in the valley downstream and in the reservoir. In order to proceed in this way, the user first removes the dam from the DEM by restoring the elevation of the cells representing the dam to the original terrain elevation.

On this DEM, the user then defines the dam as a straight line representing the dam axis (Figure5) and specifies the crest elevation and width. The end points of the dam axis are entered either by directly clicking on the screen or typed in manually as coordinates. The model interprets the cells in the rectangular area defined by the axis and the width as dam cells (Figure 5). The preprocessor toolbar (Figure 4) provides menu items for defining dams and their breaching sequences. Figure 6 shows the dialog box for defining the dam. After reading the input data CCHE2D-FLOW first raises the elevation of the cells in this rectangular to the specified crest elevation. The elevation of these cells is then modified automatically to create a breach based on the choice made by the user: 1) Sudden complete removal choice causes the program to restore all dam cells to their original terrain elevation when the specified time is reached; 2) Gradual breaching, which is defined by a sequence of breach profiles written in an ASCII file, causes the program to manipulate the elevation of the dam cells according to the specified sequence.

Quadtree Local Mesh Refinement to Improve Definition of Breach Geometry

The accuracy of the breaching described in the previous subsection depends on the computational cell size. The CCHE2D-FLOOD model includes local mesh refinement using a quadtree data structure (Figure 8). The dam cells are refined to a specified level by ensuring a gradual passage to the original cell size. Since smaller cells require a smaller time step, the computations start at the leaves of the quadtree proceeds to higher levels. When computing the cells at a given level, the higher-level cells are frozen in time. The computational burden due to refinement remains reasonable for practical cases. Detailed information is available in McGrath et al. (2010).

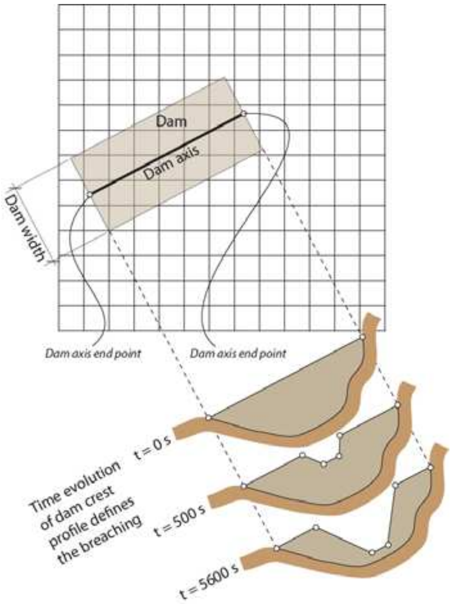
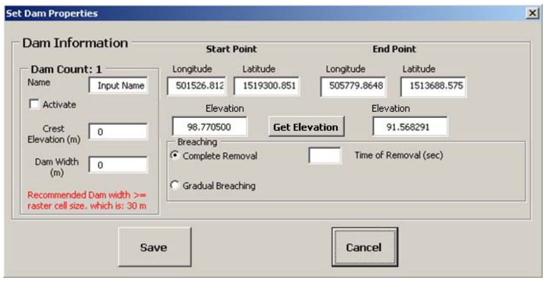


Fig. 5 Defining a dam and the associated breaching sequence.



Set Dam Properties

Dam Information

Dam Count: 1

Name: Input Name

☐ Activate

Crest Elevation (m): 0

Dam Width (m): 0

Recommended Dam width >= raster cell size. which is: 30 m

Start Point: Longitude 501526.812, Latitude 1519300.851, Elevation 98.770500

End Point: Longitude 505779.8648, Latitude 1513688.575, Elevation 91.568291

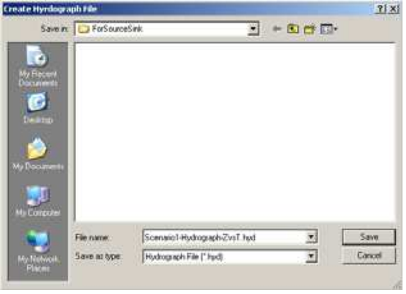
Breaching

● Complete Removal

Time of Removal (sec):

○ Gradual Breaching

Fig. 6 Dialog box for defining a dam.



Create Hyrdograph File

Save in: ForSourceSink

File name: Scenario1-Hydrograph-ZvsT.hyd

Save as type: Hydrograph File (*.hyd)

Fig. 7 Dialog box for specifying the file containing sequence of breach profiles.

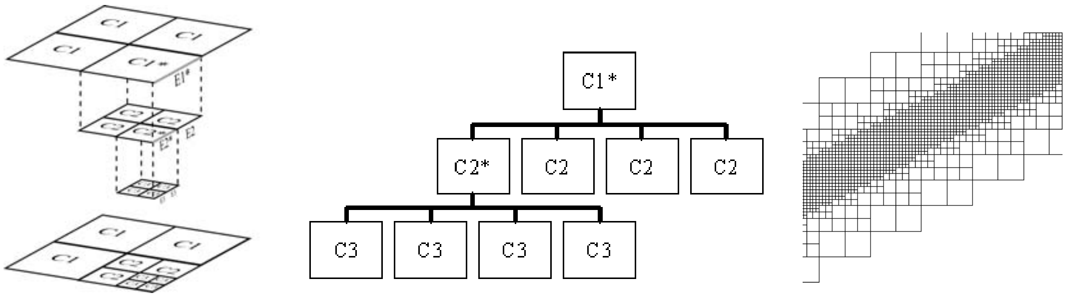


Fig. 8 Quadtree mesh refinement (left) and the associated tree structure (middle) is used to locally refine the cells under the dam and in the area around the dam.

Sources and Sinks to Represent Various Flow Conditions

Controlled releases from outlets, flow through culverts or bridges, which cannot be directly modeled in a 2D numerical model. In CCHE2D-FLOOD these can be modeled by assigning certain areas to act as sources and sinks, through which the water is added into or extracted from the computational domain. The operation of sources and sinks can be controlled in two ways:

- Source or sink is controlled by a tabular function defining the discharge as a function of time. In this case the behavior of the area is defined by the function. The area becomes a source when the discharge is positive and a sink when discharge is negative. The tabular function has to be prepared by the user based on the properties of the structure modeled.

- Source or sink discharge is computed by model using a specified mathematical function which takes into account the difference between the water level elevations in the computational domain and a user-specified outside elevation. Available types are:
    - The discharge of orifice type sink or source is controlled by the generic function $Q_{s/s} = \Delta t\ \alpha\ (\Delta H)^{\beta}$ where Δt is the time step, ΔH the elevation difference. User-defined parameters α and β are based on the diameter of the orifice and its coefficient of discharge. The user also specifies the elevation of the orifice.
    - The discharge of weir type sink or source is controlled by the generic function $Q_{s/s} = \Delta t\ \alpha \sqrt{1-[(z_{ds}-z_w)/(z_{us}-z_w)]^2}\ (\Delta H)^{\beta}$ where z_{us} and z_{ds} are the upstream and downstream water surface elevations at user specified points of the computational domain, z_{us} the elevation of the weir crest. User-defined parameters α and β are based on weir characteristics and its coefficient of discharge.

Source and sink areas can be paired to act in tandem for transferring the discharge from the sink area to the paired source area. The direction of the flow depends on the water surface elevations of the paired areas. All operation related to definition of source and sink areas are carried out easily using the menus provided in the preprocessor toolbar (Figure 4).

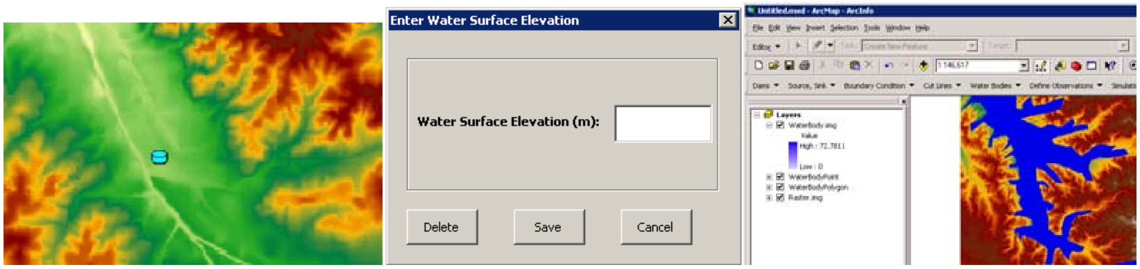


Fig. 9 Definition of initial water bodies. User selects appoint in the area to be filled with water (left) and specifies the water surface elevation (middle). The resulting initial water body is computed by the program and displayed as a raster layer (right).

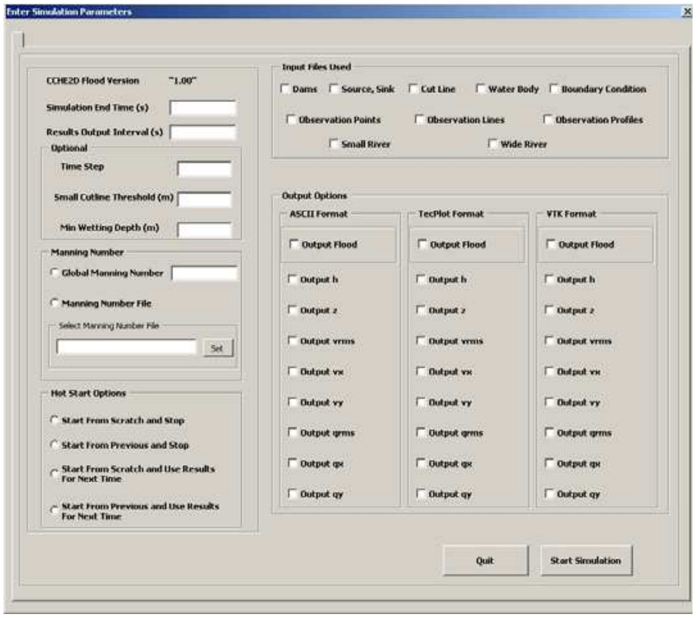
Definition of Initial Water Surfaces

The preprocessor provides a special <Water Bodies> menu (Figure 4) on the toolbar for defining initial water bodies such as reservoirs, lakes, etc. Referring to Figure 9, the user first selects <Define Water Body "Point"> from the menu and clicks at any point inside the water body to be created. The water surface elevation is specified by selecting <Set Water Surface Elevation>. When <Create Water Body> is selected the preprocessor creates the raster file of water depths by taking into account the topography, the cut-cell lines, dams and DEM boundaries. The user can also draw a polygon, and initialize the water depth of the cells inside the polygon.

Definition of Boundary Conditions

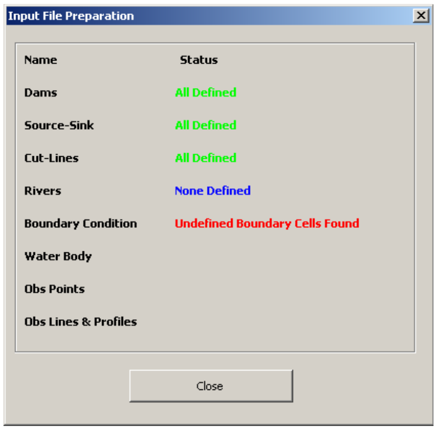
In CCHE2D-FLOOD computational domain is padded by a single line of boundary cells (Figure 1). The preprocessor provides a special menu (Figure 4) to assist the user to assign different types of boundary conditions to these boundary cells. The available boundary condition types are:

- *Non reflecting open boundary* cells are assigned an elevation extrapolated from the two nearest cells. The water surface is also assigned by extrapolation from two nearest cells. The water surface elevation in the boundary cell cannot be higher than the interior cell to prevent backflow. The case of negative extrapolated depth is handled specially.
- *Fully reflecting closed boundary* cells are simply assigned the same bottom and surface elevations as the neighboring interior cell.
- *Boundary defined by a tabular function* comes in three types: 1) discharge as a function of time; 2) water surface as a function of time; 3) stage-discharge curve. The preprocessor provides a menu to prepare or import the associated tabular functions (Figure 4).



CCHE2D Flood Version "1.00"

Fig. 10 Dialog box to enter simulation parameters.



Name	Status
Dams	All Defined
Source-Sink	All Defined
Cut-Lines	All Defined
Rivers	None Defined
Boundary Condition	Undefined Boundary Cells Found
Water Body	
Obs Points	
Obs Lines & Profiles	

Fig. 11 Program verified that all required data files exists and warns the user if there is missing information.

Definition of Observation Points, Lines and Profiles

The preprocessor has a special menu (Figure 4) to assist the user to define observation points, lines and profiles for recoding the time variation of various simulated variables for detailed analysis. At observation points the program records depth and velocity components at a specified frequency. The observation lines are used to monitor the discharge (both positive and negative) crossing the line. Observation profiles provide the depth and tangential velocity along a specified polyline. The recorded information is written into files that can be imported into a spreadsheet.

Compilation of the Input File for CCHE2D-FLOOD

Once the scenario definition is completed by defining dams, sources and sinks, cut-lines, one-dimensional rivers, boundary conditions, water bodies, and observation points, lines and profiles, the user clicks on the <Simulation Set Up> and enters the simulation parameters in the dialog box shown in Figure 10. The preprocessor then checks all given information (Figure 11) and compiles the input files for CCHE2D-FLOOD; which can be directly run from the GUI.

CONCLUSIONS

The additional features were implemented in the CCHE2D-FLOOD model, developed by the National Center for Computational Hydroscience and Engineering, the University of Mississippi. These features enable the user to prepare and simulate realistic scenarios in compliance with current engineering standards. A preprocessor embedded in ArcGIS software provides a user-friendly graphical user interface to easily define scenarios, and automatically compile input files needed by the CCHE2D-FLOOD. This model is now in use by various federal and state agencies.

ACKNOWLEDGMENTS

This research was funded by the Department of Homeland Security-sponsored Southeast Region Research Initiative (SERRI) at the U.S. Department of Energy's Oak Ridge National Laboratory.

REFERENCES

Altinakar, M.S., Matheu, E.E. and McGrath, M.Z. 2009a. New Generation Modeling and Decision Support Tools for Studying Impacts of Dam Failures. *Proc. of ASDSO Dam Safety 2009 Annual Conferenc*e, Sep. 27-Oct. 1, 2009, Hollywood, FL.

Altinakar, M.S., McGrath, M.Z., Ozeren, Y. and Miglio, E. 2009b. Two-Sided Cut-Cell Boundary Method for Simulating Linear Terrain features and 1D Stream Flows on a 2D Rectangular Mesh. *Proc. of the 33rd Int. IAHR Biennial Congress*, Aug. 9-14, 2009, Vancouver, Canada.

Altinakar, M.S., McGrath, M.Z., Ozeren, Y. and Miglio, E. 2009c. Representation of Linear Terrain Features in a 2D Flood Model with Regular Cartesian Mesh. *Proc. of the 2009 World Environmental & Water Res. (EWRI) Congress*, ASCE, May 16-23, 2009, Kansas City, MO.

Altinakar, M.S., McGrath, M.Z., Singh, J. and Ramalingam, V.P. 2009d. Coupled 1D-2D Simulations for Dam and Levee Breach Floods Using Cut-Cell Method and GIS-Based Consequence Analysis. Invited Paper, *Proc. of the Int. Symp. on Disaster Prevention, Recovery and Revitalization*, Oct 16-18, 2009, Nagaoka, Japan.

Altinakar, M.S., McGrath, M.Z., Ramalingam, V.P. and Omari, H. 2010. 2D Modeling of Big Bay Dam Failure in Mississippi: Comparison with Field Data and 1D Model Results. Accepted for publication in the *Proc. of the Int. Conference on Fluvial Hydraulics (River Flow 2010)*, Sep. 8-10, Braunschweig, Germany.

Ghias, R., Mittal, R. and Dong, H. (2007). "A sharp interface immersed boundary method for compressible viscous flows." J. Comput. Phys., 225, pp.528-553

McGrath, M.Z., Altinakar, M.S., and Miglio, E. 2010. Representation of Dam-Breach Geometry on a Regular 2-D Mesh Using Quadtree Local Mesh Refinement. Accepted for publication in the *Proc. of the Int. Conference on Fluvial Hydraulics (River Flow 2010)*, Sep. 8-10, Braunschweig, Germany.

Miglio, E. and Altinakar, M.S. and Tayfur, G. 2008. Representation of Linear Terrain Features in 2D Free Surface Models using Cut-Cell Boundary Method. *Proc. of River Flow 2008, Int. Conference on Fluvial Hydraulics*, Cesme, Izmir, Turkey, Sep. 3-5, 2008

Ying, X., Khan, A. A. and Wang, S.S.Y. (2004). "An Upwind Conservative Scheme for Saint-Venant Equations." Journal of Hydraulic Engineering. Vol.130, No.10, pp. 977-987, 2004.