Advances in Water Resources 68 (2014) 42–61

Contents lists available at ScienceDirect

Advances in Water Resources

journal homepage: www.elsevier.com/locate/advwatres

Mesh type tradeoffs in 2D hydrodynamic modeling of flooding with a Godunov-based flow solver

Byunghyun Kim [a,b], Brett F. Sanders [a,b,*], Jochen E. Schubert [a,b], James S. Famiglietti [a,b,c]

[a] University of California Center for Hydrologic Modeling, Irvine, CA 92697, USA
[b] Department of Civil and Environmental Engineering, University of California, Irvine, CA 92697, USA
[c] Department of Earth System Science, University of California, Irvine, CA 92697, USA

ARTICLE INFO

Article history:
Received 2 July 2013
Received in revised form 30 January 2014
Accepted 23 February 2014
Available online 16 March 2014

Keywords:
Flooding
Finite volume model
Mesh design
Unstructured grid
Cartesian grid
Porosity

ABSTRACT

The effect of mesh type on the accuracy and computational demands of a two-dimensional Godunov-type flood inundation model is critically examined. Cartesian grids, constrained and unconstrained triangular grids, constrained quadrilateral grids, and mixed meshes are considered, with and without local time stepping (LTS), to determine the approach that maximizes computational efficiency defined as accuracy relative to computational effort. A mixed-mesh numerical scheme is introduced so all grids are processed by the same solver. Analysis focuses on a wide range of dam-break type test cases, where Godunov-type flood models have proven very successful. Results show that different mesh types excel under different circumstances. Cartesian grids are 2–3 times more efficient with relatively simple terrain features such as rectilinear channels that call for a uniform grid resolution, while unstructured grids are about twice as efficient in complex domains with irregular terrain features that call for localized refinements. The superior efficiency of locally refined, unstructured grids in complex terrain is attributable to LTS; the locally refined unstructured grid becomes less efficient using global time stepping. These results point to mesh-type tradeoffs that should be considered in flood modeling applications. A mixed mesh model formulation with LTS is recommended as a general purpose solver because the mesh type can be adapted to maximize computational efficiency.

1. Introduction

Flood inundation models predict the spatial distribution of flood depths and velocities, or flooding intensity [27]. This defines the severity of the hazard and associated impacts such as threats to public safety, potential for monetary losses, disruptions to critical lifelines (water supply, sanitary, power and transportation systems), and general disruptions of commerce [82]. Flood inundation models can be used in a forecasting mode to support emergency management, and in a planning mode to identify the most effective risk reduction measures through comparative analysis of the socio-economic and environmental consequences of each alternative [1,27]. Monetary losses are primarily a function of flood depth and duration [48,56,71], while structural damage is also a function of flood velocity and the associated flood inertial momentum flux per unit width which scales as $\rho h V^2$ where ρ is the fluid density, h is the fluid depth, and V is the fluid velocity [31,43,44,56].

Depth-averaged shallow-water models offer an excellent foundation for flood prediction even with simplistic turbulent closures that lump all momentum losses into a resistance parameter [6]. Floods occurring on steep topography such as alluvial fans, with dam-break and levee-break flooding, and with coastal flooding from tsunami and storm surge transfer significant inertial fluxes and transition between a supercritical ($Fr > 1$) and subcritical ($Fr < 1$) state, where $Fr = V/(gh)^{1/2}$ represents the Froude number. Consequently models of these flows require a full momentum balance that accounts for local acceleration, gradients in inertial fluxes (convective acceleration), pressure gradients, gravitational effects, and friction. On the other hand, on relatively flat topography, flood flows are generally subcritical and the convective acceleration can sometimes be ignored [2,6,23]. This can be exploited to yield a model capable of relatively fast execution [22], but maintaining a complete momentum balance yields a more versatile model applicable to any possible Fr including flows with shocks [23,75].

Godunov-type models are tailored to high-inertia floods because of approximate Riemann solvers that account for transcritical flows with shocks [37,75], and the literature presents Godunov-type

* Corresponding author at: Department of Civil and Environmental Engineering, University of California, Irvine, CA 92697, USA. Tel.: +1 949 824 4327; fax: +1 949 824 3672.
E-mail address: bsanders@uci.edu (B.F. Sanders).
URL: http://sanders.eng.uci.edu (B.F. Sanders).

http://dx.doi.org/10.1016/j.advwatres.2014.02.013
0309-1708/

modeling studies of dam-break floods [9,30,53,67,77,81], coastal floods [25,32] and urban floods [39,57,66,67,76]. The success of Godunov-type flood models can be attributed to approximate Riemann solvers [37,75] which are also embedded in discontinuous Galerkin finite element schemes [4,26,45] and Boussinesq models that account for non-hydrostatic flow effects [47].

Godunov-type models have generally assumed either a structured mesh of quadrilateral cells [3,13,29,35,42,86] or an unstructured mesh of triangular cells [7,15,41,70,84]. The latter mandates greater overhead to track the neighborhood of data around each cell, and makes it more challenging to compute gradients in the solution because data points do not fall on a regular grid [7,41], but the unstructured mesh is very appealing for the ease with which meshes can be generated and tailored to the unique geometry of application sites and the ability to locally refine the mesh around areas of interest [11,21,54,57,66,67,76]. Adaptive mesh approaches also allow for local refinements, and take this a step further by adjusting refinements on the fly according to flow conditions [46,52]. Cut-cell models represent a variant of quadrilateral mesh models wherein any particular quadrilateral can be bisected so as to better constrain the mesh to the site geometry [17,18]. A few studies have also used a mixed-mesh Godunov-type shallow-water model, i.e., a model that can utilize a mesh of either triangular cells, quadrilateral cells, or any combination of triangles and quadrilaterals [1,19,57].

When designing a mesh, the goal is achieve the highest degree of accuracy for a given computational cost, our alternatively, the minimum computational cost to achieve a specified level of accuracy. Several factors contribute to accuracy of flood models including: (1) structural model errors, (2) input data errors, and (3) numerical errors. Structural model errors stem from limitations of the governing equations, in this case the depth-averaged shallow-water equations, such as the assumption that pressure is hydrostatic, the velocity distribution is unidirectional (not stratified or skewed), and the bed is fixed (not erodible). Structural model errors will depend on whether the model solves the full shallow-water equations or simplified forms of the 2D equations [2,22], uses a coupled 1D/2D approach for channel flows [5,12,49,83], or introduces a sub-grid topographic model [60] or sub-grid obstruction model [38,65]. Input data errors include errors in initial conditions, boundary conditions, and parameters such as the resistance parameters and, most importantly, elevation data. Numerical errors are linked to the chosen numerical method and include errors in the spatial and temporal discretization. For example, upwind schemes promote a diffusive error that can smear out sharp fronts (e.g., hydraulic jumps) if an overly coarse discretization is used.

Mesh design affects both input data errors and numerical errors. Topographic heights and resistance parameters are sampled at the vertices (or cell centers) of the mesh, and thus mesh design controls sampling. In areas of high topographic variability, a fine mesh may be required to resolve important flow paths and thus minimize input data errors. Similarly, a relatively fine mesh can be used to resolve sharp flow features such as a hydraulic jump and thus minimize numerical errors. A recent review of flood inundation modeling indicates that uncertainties in topographic data and hydrologic data remain the primary source of uncertainty in flood predictions [6], and a St. Francis dam-break modeling study concluded that reduction of topographic errors and numerical errors were both important for model accuracy [10]. Research also indicates that model structure can affect the sensitivity of model predictions to topographic errors [2].

Mesh design also drives computational cost, which for an explicit finite volume model scales in proportion to the number of cells N_C and the number of time steps N_T as follows,

$$\mathcal{C} \sim k_1 N_C N_T \tag{1}$$

where k_1 is a factor that depends on the numerical scheme, i.e., the spatial and temporal discretization and solution update procedure. The number of time steps depends on the mesh resolution as a result of the CFL condition given by [74],

$$\alpha = \frac{\lambda_{\max}\Delta t}{\Delta s} \leqslant \alpha_{max} \tag{2}$$

where α is the Courant number, α_{max} is the maximum Courant number for stability, Δt is the time step, Δs is the grid size, and $\lambda_{\max}$ is the maximum wave speed here given by $V + (gh)^{1/2}$. The stability limit α_{max} depends on the numerical method, and Δs is not easily defined for unstructured grids [74]. Nevertheless, computational costs will generally increase with mesh resolution as follows,

$$\mathcal{C} \sim k_2 \Delta s^{-3} \tag{3}$$

because $N_C \sim \Delta s^{-2}$ and $N_T \sim \Delta s^{-1}$ based on the CFL condition (Eq. (2)), where k_2 is a factor that depends on the numerical scheme.

Clearly, computational costs and accuracy are at odds with respect to mesh resolution. But what is the role of mesh type? Flood modeling domains typically include regions where a fine resolution is required to minimize topographic errors, such as a channel or raised embankment that constrains the spreading of flood water, and other regions where added resolution is not necessary, such as a floodplain. In this case, an unstructured mesh appears to be advantageous compared to a Cartesian grid because a fine resolution can be used selectively, thus minimizing N_C and reducing $\mathcal{C}$. On the other hand, when topographic heights are uniformly gridded as a Digital Terrain Model (DTM), and flow modeling proceeds at the resolution of the DTM, Cartesian grids appear to be advantageous because fewer cells can be used. For example, when each cell of a Cartesian grid is simply divided diagonally to make a triangular grid, $\mathcal{C}$ is magnified because both N_C and N_T are increased. While the added resolution of the triangular grid may help to reduce numerical error, there is no difference in the topographic error between the two grids so overall accuracy is unlikely to differ. To date, the effect of mesh type on the computational effort and accuracy of Godunov-type flood models has not been critically examined. With increasing urbanization and climate changes that threaten more extreme flooding, the need to model flooding in detail is growing on several fronts (e.g., forecast systems, risk reduction programs) and therefore a better understanding of how to develop computationally efficient models is needed.

1.1. A study of mesh-type tradeoffs

This paper presents a study on how computational costs and accuracy are affected by mesh type using Godunov-type flood models. The study is focused on the central question of which mesh type achieves the most accurate prediction (smallest errors) for the same computational expense, i.e., the maximum computational efficiency. Alternatively, the question can be viewed as which mesh design achieves a desired level of accuracy for the minimum computational effort. A mixed-mesh Godunov-type finite volume scheme is used to compare several mesh designs including Cartesian grids, unstructured grids of triangular cells, and mixed-meshes of triangular and quadrilateral cells. Further, the effect of mesh design on accuracy and computational cost is isolated by using the same computational engine in all cases. The eliminates biases that would otherwise result from different solvers and/or model structures.

The remainder of the paper proceeds as follows: Section 2 presents the governing equations and a brief description of the numerical method. A detailed presentation of the model is included as an Appendix. Section 3 presents applications ranging from channel flow problems to large scale flooding problems, and including idealized problems with exact solutions, laboratory-scale