Real-time simulation of large-scale floods

Q Liu[1], Y Qin[1, 3], G D Li[1], Z Liu[2], D J Cheng[2] and Y H Zhao[2]

[1] Key Laboratory of the North-West Water Resources and Ecology Environment, Ministry of Education, Xi'an University of Technology, Xi'an 710048, China;
[2] Tianjin Water Authority, 210 Weidi Road, Hexi District, Tianjin 300074, China

E-mail: 13571991500@126.com

Abstract. According to the complex real-time water situation, the real-time simulation of large-scale floods is very important for flood prevention practice. Model robustness and running efficiency are two critical factors in successful real-time flood simulation. This paper proposed a robust, two-dimensional, shallow water model based on the unstructured Godunov-type finite volume method. A robust wet/dry front method is used to enhance the numerical stability. An adaptive method is proposed to improve the running efficiency. The proposed model is used for large-scale flood simulation on real topography. Results compared to those of MIKE21 show the strong performance of the proposed model.

1. Introduction

In recent years, flood inundation mapping projects have been widely implemented in China. The flood inundation maps of the maximum water depth, the flood arrival time, and the flood duration are drawn according to many typical cases of hydrology condition and dyke burst. Those maps should be very useful for flood prevention, especially for emergency planning. However, the real-time water situation and hydraulic engineering states are very complex and changeable, so the flood inundation maps of typical cases might be quite different with the real flood situation, and the real-time simulation of large-scale floods is very important for flood prevention practice.

Many flood simulation models have been developed in recent years, including the 1D [1], 2D [2-4], and 1D-2D coupled [5,6] hydrodynamic models. A number of 2D flood simulation models used the 2D shallow water equations as the governing equations and adopted the Godunov-type finite-volume methods for numerical resolution. Due to the well conservation property, the Godunov-type finite-volume method is more suitable than the finite-difference method for flood simulation with wetting and drying on complex topography. Although finite-volume based flood simulation models have been researched for many years, methods published were mainly focused on the model accuracy and numerical stability [2-4]. Some parallel models [7] were proposed for faster computation. Parallel models should achieve real-time simulation of large-scale floods, but they need an expensive computation server, which also requires great power consumption.

According to the two critical factors of model robustness and running efficiency, this paper proposed a robust, two-dimensional, shallow water model based on the unstructured Godunov-type finite volume method. A robust wet/dry front method is used to enhance the numerical stability. An adaptive method is proposed to improve the running efficiency.

2. Numerical model

The 2D shallow water equations are used as the governing equations [6],

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = \mathbf{S} \tag{1}$$

in which,

$$\mathbf{U} = \begin{bmatrix} h \\ hu \\ hv \end{bmatrix}, \quad \mathbf{E} = \begin{bmatrix} hu \\ hu^2 + g(h^2 - b^2)/2 \\ huv \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} hv \\ huv \\ hv^2 + g(h^2 - b^2)/2 \end{bmatrix},$$

$$\mathbf{S} = \mathbf{S}_0 + \mathbf{S}_f = \begin{bmatrix} 0 \\ g(h+b)S_{0x} \\ g(h+b)S_{0y} \end{bmatrix} + \begin{bmatrix} 0 \\ -ghS_{fx} \\ -ghS_{fy} \end{bmatrix}$$

where h is the depth; u and v are the x- and y-velocity; b is the bottom elevation; S_{0x} and S_{0y} are the x- and y-bed slopes, $S_{0x} = -\partial b/\partial x$ and $S_{0y} = -\partial b/\partial y$; g is the gravity acceleration; S_{fx} and S_{fy} are the friction terms estimated by Manning formulae:

$$S_{fx} = \frac{n^2 u \sqrt{u^2 + v^2}}{h^{4/3}} \quad S_{fy} = \frac{n^2 v \sqrt{u^2 + v^2}}{h^{4/3}} \tag{2}$$

The discretization equation can be obtained by using Green's theorem to integrate equation (1), which is given as

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{\Delta t}{\Omega_i} \sum_{k=1}^{3} \mathbf{F}_k(\mathbf{U}_L, \mathbf{U}_R) \cdot \mathbf{n}_k L_k + \frac{\Delta t}{\Omega_i} \mathbf{S}_i \tag{3}$$

where the subscript i is the cell number; $\mathbf{U}_i$ is the cell-averaged conservative vector; n is the time step; k and L is the edge number and length respectively; $\mathbf{n}$ is the unit outward vector normal to the edge; Ω is the cell area; $\mathbf{S}_i$ is the source approximation; $\mathbf{F}$ is the outward normal flux vector; $\mathbf{U}_L$ and $\mathbf{U}_R$ are the local left and right states for Riemann problem respectively; Δt is the time step.

The MUSCL-Hancock scheme presented in [6] is also used in this paper for numerical computation. Model details can be found in [6], and this paper only presents improvement on numerical stability and running efficiency.

In [6], the VFRs are used to calculate water level, and cells are classified as wet-cell and dry-cell, based on the nodal water depth. Only the continuity equation is updated for dry cells, whereas the continuity and momentum equations are simultaneously updated for wet cells. Those should compose a robust method for wetting/drying treatment. However, since the momentum equation is not updated for dry cells, the flow velocities are set to zero to the cells at the wet/dry fronts, which would be inconsistent with real flow condition. To deal with this problem, a new wet/dry front method is proposed for reconstruction to the cells at the wet/dry fronts, given as,

$$h_{i,k}^{\text{Rec}} = \begin{cases} 0 & \text{if} \quad \eta_i \le b_{i,k}^{\min} \\[2mm] \dfrac{(\eta_i - b_{i,k}^{\min})^2}{2(b_{i,k}^{\max} - b_{i,k}^{\min})} & \text{if} \quad b_{i,k}^{\min} < \eta_i \le b_{i,k}^{\max} \quad (k = 1, 2, 3) \\[3mm] \eta_i - \dfrac{b_{i,k}^{\min} + b_{i,k}^{\max}}{2} & \text{if} \quad \eta_i > b_{i,k}^{\max} \end{cases} \tag{4}$$

$$u_{i,k}^{\text{Rec}} = u_i \quad v_{i,k}^{\text{Rec}} = v_i \quad (k = 1, 2, 3) \tag{5}$$

where η_i is the water surface elevation; $b_{i,k}^{\min}$ and $b_{i,k}^{\max}$ are the minimum and maximum bed elevations of the edge. When the numerical fluxes are computed using the reconstructed values, the continuity and momentum equations are simultaneously updated for wet-cells, as well as the cells at the wet/dry fronts.

In [6], a fixed time step is used for 1D-2D coupled computation, since the 1D hydrodynamic model, based on the finite-difference method, should adopt a fixed time step for computation. However, the fixed time step should be very small, to preserve the stability condition during the whole simulation. This would markedly decrease the running efficiency of the 2D model. To deal with this problem, an adaptive method is proposed to improve the running efficiency, given as,

$$\Delta t = C_r \cdot \min_{i,k}\left[\left(\frac{\Omega}{(|u_\perp|+\sqrt{gh})_k L_k}\right)_i\right] \quad i = 1,2,\cdots,N; k = 1,2,3 \qquad (6)$$

where Δt is the time step; C_r is the Courant number, and $C_r = 0.95$; $u_\perp$ and h are the average of the reconstructed value; N is the total number of cells. The adaptive time step should improve the running efficiency for dyke break flood simulation, since the time step would be very small when dyke breach occurs, but would be larger when the flow velocity is small.
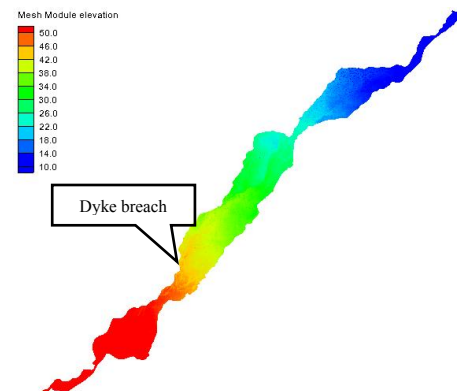
3. Case study: a real flood simulation in Wei River Basin

3.1. Computational mesh
The study area is the Wei River Basin, which is triangulated by 179296 grids, and the length of edges is around 200m-250m. The average area of grids is 0.053km2. The area of the computation domain is about 9503 km2, which is a large-scale field. Figure 1 shows the local computational mesh. Figure 2 shows the bed elevation.
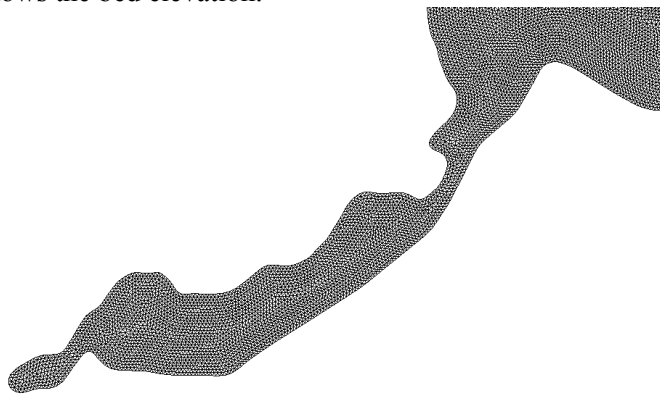

Figure 1. The local computational mesh.   **Figure 2.** The bed elevation.

3.2. Boundary condition
A hypothetical dyke breach and the position are shown in Figure 2. The discharge across the breach is shown in Figure3. The peak discharge is 496m³/s, and the flooding time is about 263h. To simulate the whole process of flooding and recession, the computation time is set to 878h, and the discharge across the breach is 0 during 264h-878h.
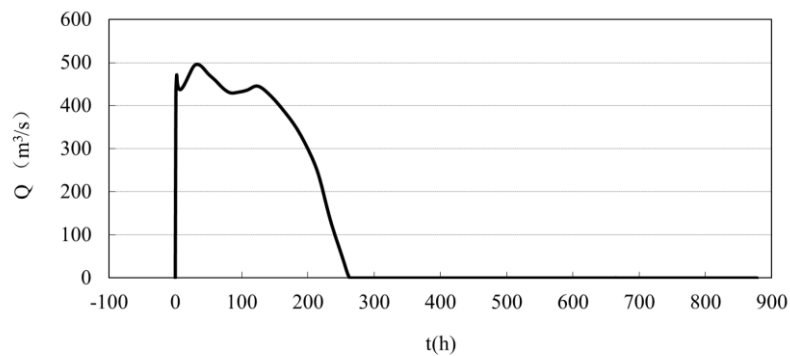
**Figure 3.** The discharge across the breach.

3.3. Simulated results

Figure 4 shows the distribution of water depth at different times, which gives a reasonable process of flood routine.
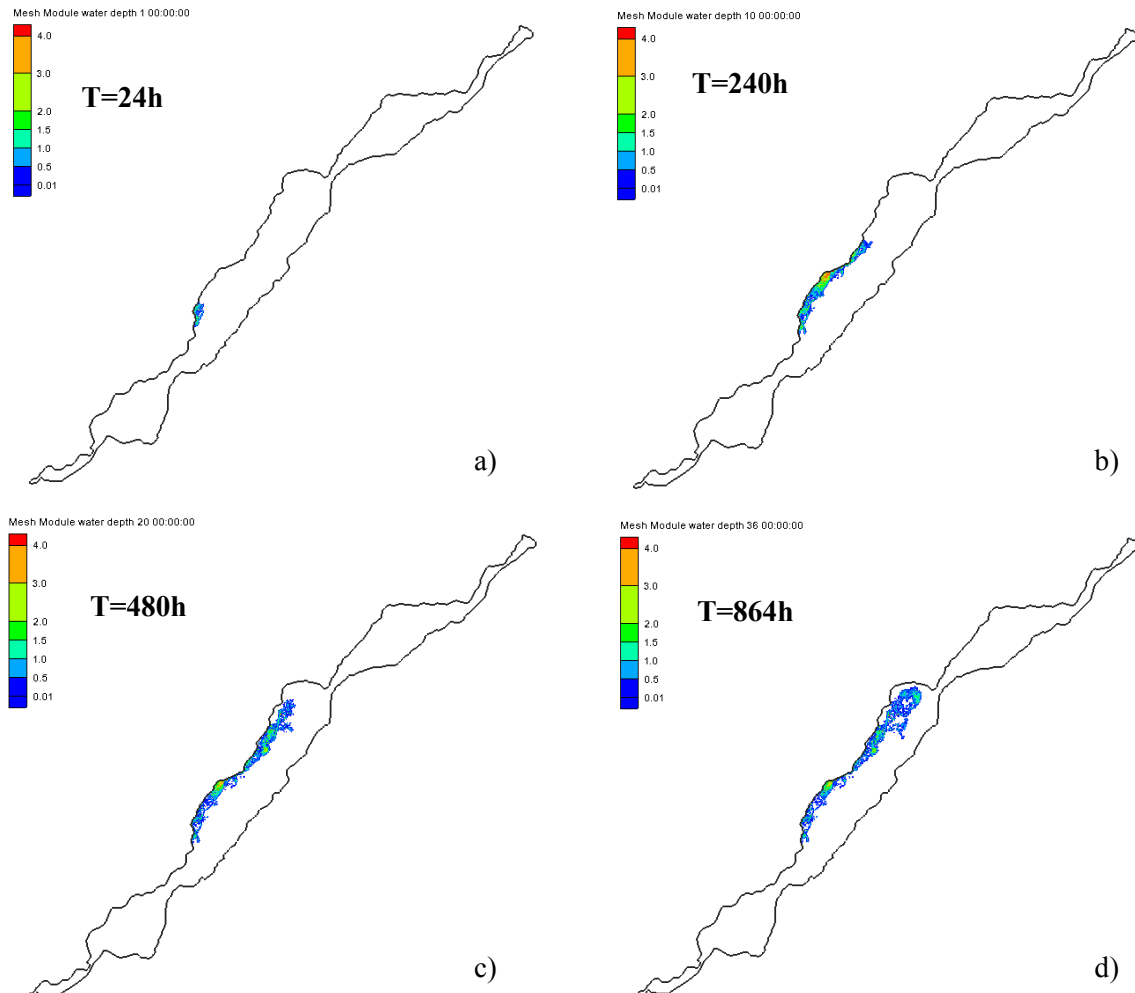
Figure 4. The distribution of water depth at different times.

Figure 5 shows the mass conservation result, which validates the well conservation property of the proposed model. During the simulation, the water volume from breach is almost equal to the water volume of cells, and the relative error is about 0.017% at the end. The mass error is very small, and it should not induce a falsity flow.
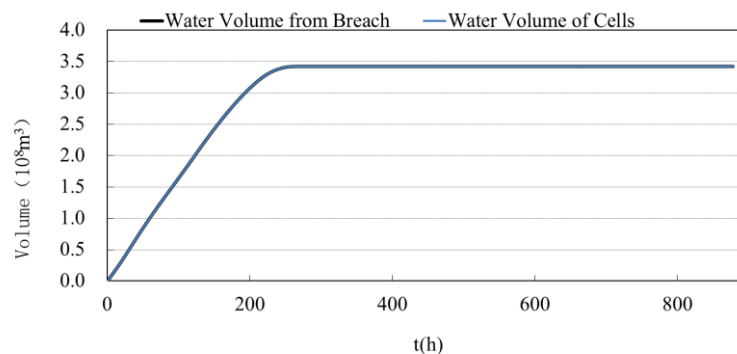
Figure 5. The mass conservation result.

Table 1 gives the comparison of the final flooded area between the proposed model and the MIKE 21 model. Results show the good agreement with MIKE 21.

Table 1. The comparison of the final flooded area (km^2).

Model	Flooded area	0.05-0.5m	0.5-1m	1-2m	2-3m	>=3m
MIKE 21	864.26	370.35	234.5	200.34	50.14	8.93
The proposed model	856.28	364.62	241.19	193.46	47.03	9.98
Relative error(%)	-0.92%	-1.55%	2.85%	-3.43%	-6.20%	11.76%

The flood inundation maps of the maximum water depth, and the flood arrival time are also comprised in MIKE 21 results, see figure 6 and figure 7. Results also show the good agreement with MIKE 21.
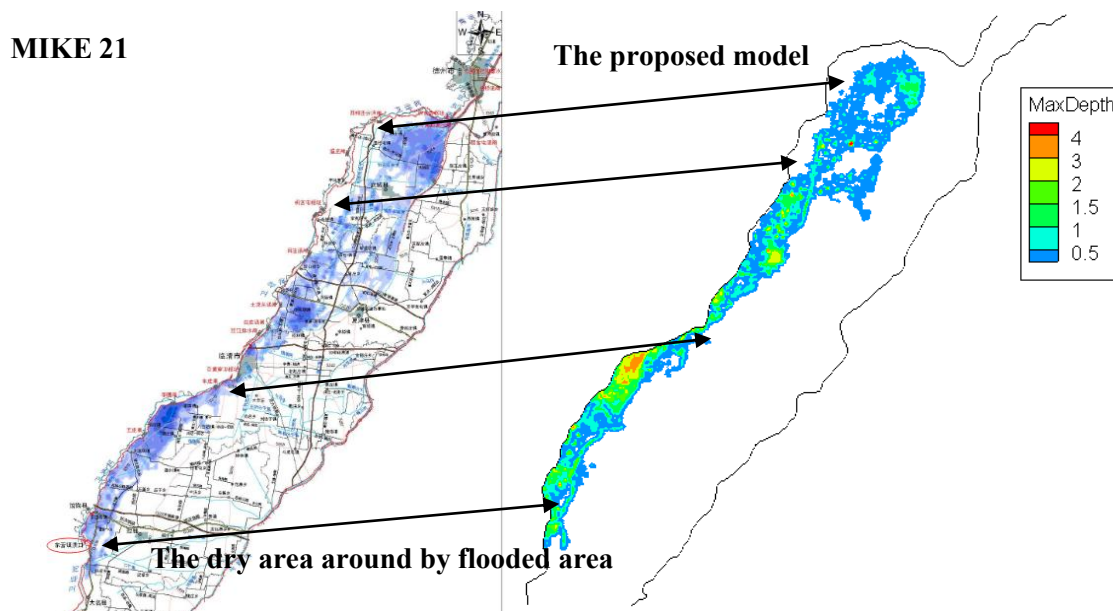
Figure 6. The flood inundation maps of the maximum water depth.

MIKE 21   **The proposed model**
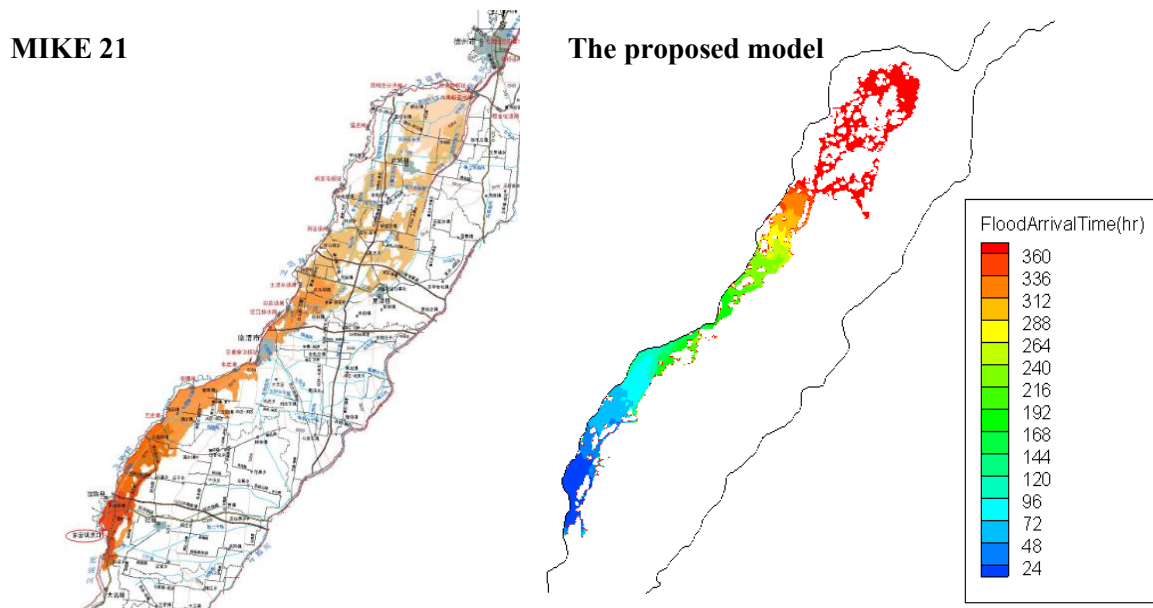
Figure 7. The flood inundation maps of the flood arrival time.

　　The running time of the proposed model is 1.7h, which shows the computational efficiency of the proposed model. In this study, a 22-day flooding process can be simulated in one hour, so it can be concluded that the proposed model is useful for real-time simulation of large-scale floods

4. Conclusions

According to the two critical factors of model robustness and running efficiency, this paper proposed a robust, two-dimensional, shallow water model based on the unstructured Godunov-type finite volume method. A robust wet/dry front method is used to enhance the numerical stability. An adaptive method is proposed to improve the running efficiency. The proposed model is used for large-scale flood simulation on real topography. Results compared to those of MIKE21 show the strong performance of the proposed model.

References

[1]   Zhu D, Chen Y, Wang Z and Liu Z 2011 Simple, robust, and efficient algorithm for gradually varied subcritical flow simulation in general channel networks *ASCE Journal of Hydraulic Engineering* **137**(7): 766-74

[2]   Song L, Zhou J, Guo J, Zou Q and Liu Y 2011 A robust well-balanced finite volume model for shallow water flows with wetting and drying over irregular terrain *Advances in Water Resources* **34**(7): 915-32

[3]   Song L, Zhou J, Li Q, Yang X and Zhang Y 2011 An unstructured finite volume model for dam- break floods with wet/dry fronts over complex topography *International Journal for Numerical Methods in Fluids* **67**(8): 960-80

[4]   Liu Y, Zhou J, Song L, Zou Q, Liao L and Wang Y 2013 Numerical modelling of free-surface shallow flows over irregular topography with complex geometry *Applied Mathematical Modelling* **37**(23): 9482-98

[5]   Blade E, Gomez-Valentin M, Dolz J, et al. 2012 Integration of 1D and 2D finite volume schemes for computations of water flow in natural channels *Advances in Water Resources* **42**: 17-29

[6]   Liu Q, Qin Y, Zhang Y, et al. 2015 A coupled 1D–2D hydrodynamic model for flood simulation in flood detention basin *Natural Hazards* **75**(2):1303-25

[7]   Sanders B F, Schubert J E and Detwiler R L 2010 ParBreZo: A parallel, unstructured grid,

Godunov-type, shallow-water code for high-resolution flood inundation modeling at the regional scale *Advances in Water Resources* **33**(12): 1456-67