Available online www.jocpr.com

Journal of Chemical and Pharmaceutical Research, 2014, 6(6):603-607

Research Article

ISSN : 0975-7384
CODEN(USA) : JCPRC5

Numerical simulation of hydraulic jump using ENO scheme

Y. L. Liu*, Y. X. Cai, W. L. Wei, K. Yang and Zh. Ma

State Key Laboratory of Eco-Hydraulic Engineering in Shaanxi, Xi'an University of Technology, Xi'an, Shaanxi, China

---

ABSTRACT

This paper is concerned with a high-resolution mathematical model for one-dimensional (1D) hydraulic jump flows. The 1D shallow water equations of Saint-Venant were solved with the essentially non-oscillatory (ENO) scheme, the finite volume method (FVM) and the total variation diminishing (TVD) Runge-Kutta-type method. This model was used to predict the hydraulic jump flows, and the numerical resolutions are in agreement with the experimental ones, which indicate that this model is fairly effective and accurate for simulating hydraulic jump flows.

Keywords: ENO scheme; FVM; hydraulic jump, numerical simulation, water equations of Saint-Venant

---

INTRODUCTION

Hydraulic jump is formed when the state of flow changes from supercritical to subcritical one. The hydraulic jump is usually accompanied by an aerated recirculating roller on the top, where intensive turbulence results in energy dissipation and the mixing of air also takes place. The hydraulic jump is treated as a macroscopically steady phenomenon for engineering purposes, and temporally averaged quantities are examined.

Mathematically, the hydraulic jump is commonly described by the shallow water equations (also named the Saint Venant equations for the 1D case). One feature of hyperbolic equations of this type is the formation of bores (i.e., the rapidly varying discontinuous flow). It is an important basis for validating the numerical method whether the scheme can capture the hydraulic jump waves accurately or not. This gives rise to an increasing interest in solving such a problem. Several finite-difference schemes that handle discontinuities effectively were used to compute open-channel flows, such as the approximate Riemann solver [1-4]. In recent years, more and more high-resolution schemes such as MacCormack, TVD and ENO schemes [5-7] are applied into numerical simulation of discontinuous problem such as the shock wave. Although TVD scheme can keep the total variation diminishing, it causes the accuracy drop to one-order at the local extreme point of smooth area. To overcome this weakness, ENO (Essentially no-oscillatory) scheme [7] is proposed, which amplifies the restriction of total variation diminishing and also allows a tiny increase of total variation, then the scheme is kept uniformly high resolution and essentially no-oscillatory. The character of ENO scheme is that it eliminates the monotonicity limit, and can adaptively choose the best interpolating point in the interpolating functions reconstructions.

Based on the above research results, the goal of the current work is to develop a mathematical model capable of dealing with hydraulic discontinuities such as steep fronts, hydraulic jump, etc. The water governing equations has been solved by the ENO scheme.

GOVERNING EQUATIONS

The 1D equations of Saint Venant are as follows [7]:

$$\partial U/\partial t + \partial F/\partial x + G = 0 \tag{1}$$

in which the variables *U*,*F*, and *Q* are defined in matrix forms as follows: $U = (A,Q)^T$; $G = (0, gAS_f)^T$; $P = P(z) = \int_{z_0}^{z} (z-\eta)B(\eta)d\eta$; $S_f = n^2 Q|Q|B^{4/3} / A^{10/3}$,where *t*=time, *x*=distance, *A*=stream cross-section area, *Q*=discharge, *G*=gravity, *P*=hydrostatic pressure on the cross-section, *z*= water level, z_0 is the lowest elevation of cross-section, $B(\eta)$ is the width when the water level is η, S_f is variable friction slope, *n* is the Manning coefficient, Figure 1 shows the definition sketch for general flows in channels.

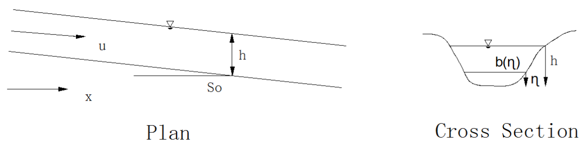

Fig. 1 Definition sketch for general flows in channels

NUMERICAL SCHEME

The numerical schemes include the numerical discretization of water equations, numerical flux splitting, ENO schemes reconstruction and time discretization [7,8].

Numerical discretization of water equations:

The equation (1) can be discretized as [8]:

$$U_i^{n+1} = U_i^n - \lambda\left(\overline{F}_{i+1/2}^n - \overline{F}_{i-1/2}^n\right) - \Delta t G_i \tag{2}$$

where $\lambda = \Delta t / \Delta x$; Δt =time step; Δx =spatial step; n represents the time layer; i represents the spatial point; $\overline{F}_{i+1/2}^n$ and $\overline{F}_{i-1/2}^n$ are the numerical fluxes consistent with flux F of equation (1).

Numerical flux splitting:

The finite-difference method (FDM) was used to solve the numerical flux, such as $\overline{F}_{i+1/2}^n$:

$$\overline{F}_{i+1/2}^n = \frac{1}{2}\left\{F\left(U_{i+1/2}^L\right) + F\left(U_{i+1/2}^R\right) - |\tilde{a}|\left(U_{i+1/2}^R - U_{i+1/2}^L\right)\right\} \tag{3}$$

where $\tilde{a}$ is the Roe average value of $a = \partial F / \partial U$ on U^L and U^R ,and is the Jacobian matrix for $a = \partial F / \partial U$. $U_{i+1/2}^L$ and $U_{i+1/2}^R$ are the values of U on the left or right hand of point $x_{i+1/2}$ respectively, which can be computed by the stencil option of ENO .The Roe velocity is as follows:

$$\tilde{a} = \frac{F\left(U_{i+1/2}^R\right) - F\left(U_{i+1/2}^L\right)}{U_{i+1/2}^R - U_{i+1/2}^L} \tag{4}$$

Similarly, $\overline{F}_{i-1/2}^n$ can be computed with same method and the values of $U_{i-1/2}^L$ and $U_{i-1/2}^R$ can also be obtained by the stencil option of ENO schemes.

ENO schemes reconstruction:

ENO schemes adaptively choose the smoother stencil by comparing the size of divided difference. To achieve k -order resolution schemes, k cell is needed when choosing the stencil. Total cells are added to $2k-1$. Assume the k stencil as follows:

$$S_r(i) = \{x_{i-r}, \cdots, x_{i-r+k-1}\}, \ r = 0, \cdots, k-1, \tag{5}$$

So k different reconstruction method to calculate $U_{i+1/2}^{L}$ or $U_{i+1/2}^{R}$ can be achieved:

$$U_{i+1/2}^{(r)} = \sum_{j=0}^{k-1} C_{r,j}\overline{U}_{i-r+j}\ ,\quad r = 0,\cdots,k-1 \tag{6}$$

where $\overline{U}_i$ is the average value of U at point x_i ,which can be written as:

$$U_{i+1/2}^{L,R} = \sum_{r=0}^{k-1} w_r U_{i+1/2}^{(r)} \tag{7}$$

Let the coefficient of the decided stencil whose magnitude of divided difference is minimal w_r = 1, the others w_r = 0, then the stencil option is finished.

If $w_r \geq 0$ and $\sum_{r=0}^{k-1} w_r = 1$, then the equations was changed to WENO(Weighted ENO) schemes.

Time discretization:

The Runge-Kutta TVD method was used to discretize the equation (1). The equation can be written as [9]:

$$\frac{\partial U}{\partial t} = L(U) \tag{8}$$

where $L(U)$ is approached by $-F(U)_x - G$.

According to the Runge-Kutta TVD schemes,

$$\begin{aligned} U^{(1)} &= U^{n} + \Delta t L\left(U^{n}\right) \\ U^{(2)} &= \frac{3}{4}U^{n} + \frac{1}{4}U^{(1)} + \frac{1}{4}\Delta t L\left(U^{(1)}\right) \\ U^{(n+1)} &= \frac{1}{3}U^{(n)} + \frac{2}{3}U^{(2)} + \frac{2}{3}\Delta t L\left(U^{(2)}\right) \end{aligned} \tag{9a,b,c}$$

RESULTS AND DISCUSSION

The test problem relates to a hydraulic jump in a wide horizontal rectangular channel of constant width. A hydraulic jump is formed whenever the flow changes from super-critical flow (Fr>1) to sub-critical flow (Fr<1), where Fr represents the Froude number. The initial flow conditions in the horizontal channel are a depth of 0.04 m and a velocity of 2.65 m/s. These flow conditions give an upstream Froude number of 4.23. A Courant number of 0.8, a roughness coefficient of 0.003, and a fine grid spacing of 0.3 m were used. As supercritical flow has an upstream control, the flow variables at the upstream node were kept equal to the initial conditions. At the downstream end a constant depth of 0.2 m was specified and held constant for all time levels.

Figure 2 is the transient depth profile. The computed results and the experimental ones are in a close agreement.

Theoretically, a hydraulic jump will occur when the upstream Froude number Fr_1, depth h_1 and downstream depth h_2 satisfy the Bélanger formula $h_2 / h_1 = \frac{1}{2}(\sqrt{8Fr_1^2 + 1} - 1)$. This can be used to check the results by the model. From the numerical results, the downstream water depth h_2=0.23m. Substitution of the upstream Froude number and depth (Fr_1=4.23, h_1=0.04m) into the Bélanger formula results in the theoretical downstream depth required for the jump as $\hat{h}_2$ =0.22m. h_2 is thus very close to the theoretical downstream depth $\hat{h}_2$. The relative error is 10%.

The close agreement between the computed results and the experimental ones shows that the proposed method is comparatively accurate.

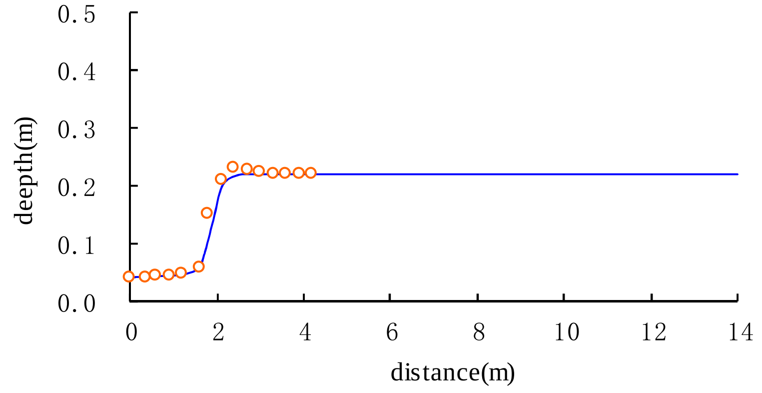

Fig.2 Hydraulic jump steady state profiles for F_r=4.23(—computed, ○experimental)

Figure 3 shows the effect of grid spacing on shock reduction. Under the same conditions that Fr_1 = 4.23, h_1 = 0.04m ,the comparisons of the computed results with Δx of 0.3m and 0.12m show that varying the grid spacing and Courant number did not alter the location of the shock front, which says that grid spacing of 0.3m has converged to steady state with enough accurate solution.

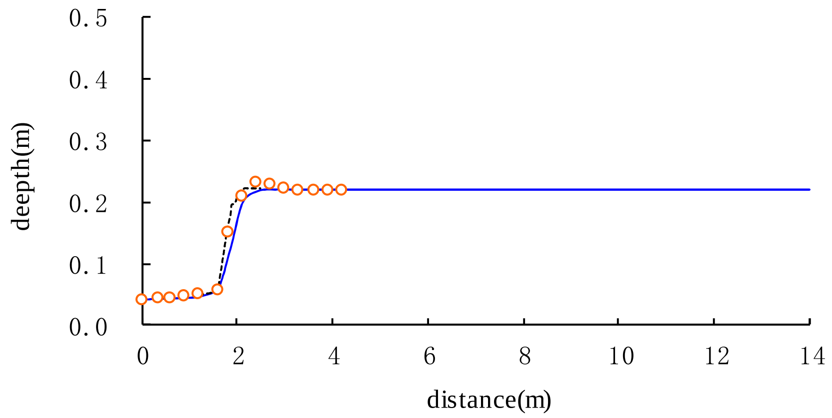

Fig.3.Effect of grid spacing on shock reduction, F_r=4.23 (—Δx=0.3,Δx=0.12, ○ experimental)

CONCLUSION

The high-resolution mathematical model has been developed for 1D shallow water equations (Saint-Venant equations) for the computation of hydraulic jump flows. The 1D shallow water equations were solved with the ENO scheme, the finite volume method (FVM) and TVD Runge-Kutta-type method. Numerical simulation for the hydraulic jump flows indicates that this model is fairly effective and accurate for simulating hydraulic jump flows. The high-resolution mathematical model can be further modified and extended to multi-dimensional hyperbolic conservation laws to effectively simulate 2D and 3D hydraulic jump flows. When utilizing the technique of boundary treatment such as a body-fitted coordinate system, the proposed model can also effectively simulate the hydraulic jump flows with complex boundaries. The results will be given in a future work.

.

Acknowledgments

Partial financial support of this study from the National Natural Science Foundation of China (Grant No.51178391), the scientific research project of Shaanxi Province (2014K15-03-05)and the special funds for the development of characteristic key disciplines in the local university supported by the central financial (Grant No: 106-00X101, 106-5X1205) and the Shanxi Province key subject construction funds.

REFERENCES

[1]K Hu , C G Mingham , D M Causon, *Int. J.Numer. Methods in Fluids*, **1998**,28,pp1241~1261.

[2]K Anastansious , C T Chan, *Int. J. Numer. Methods in fluids* , **1997**, 24,pp1225-1245.
[3]C G Mingham, D M Causon, *J. Hydr. Engrg., ASCE*, **1998**, 124, pp605-614.
[4]P Glaister, *Int. J. Numer. Methods in fluids*, **1993**, 16,pp629-654.
[5]A. Harten, S. Engquist, Osher and S. Chakravarthy, *Journal of Computational Physics*, **1987**, 71, pp231-303.
[6]A. Harten, *J Comp Phys*, **1983**,49, pp357-393.
[7]Senka Vuknovic, Luka Sopta, *Journal of Computational Physics* , **2002**,179, pp593-621.
[8]W. L. Wei, Computational hydromechanics theory with applications, 1st Edition,Xi'an, Shanxi Science and Technology Publishing House, **2010**,92-93.
[9]Y. L. Liu, YF Hong, W. L. Wei , *Journal of Chemical and Pharmaceutical Research*, **2014**, 6(4),265-269.