*J. Eng. High. Tech.*, 4(1)1-9, 2018
© 2018, TextRoad Publication
ISSN 2356-8860
**Journal of Engineering and Higher Technology (JEHT)**
*www.textroad.com*

# Studies of Stationary Shocks Behavior in Open Channel Flows in Dimensionless Forms

**Abbas Ali Ghezelsofloo**

Assistant Professor, Civil Engineering Department, Islamic Azad University, Mashhad Branch, Iran

## ABSTRACT

In this article, the behavior of fully developed stationary shocks, caused by the incidence of a supercritical flow with a cross-barrier in an open channel, is simulated. Due to abrupt changes in flow depth and velocity as well as some variability of flow regime to sub and supercritical, the conventional numerical methods are not able to capture the produced discontinuities and shocks. The numerical solution of non-linear governing shallow flow equations have been implemented by the application of a second–order Roe–TVD scheme which is known as shock capturing scheme. The obtained results from numerical experiment compared with some measured in a laboratorial setup. Next the numerical model has been calibrated by adding a virtual roughness to represent some turbulence of supercritical flow at downstream of cross-barrier with difference length in laboratory. The positions of consequence, oblique and complicated shocks reflection are simulated in percent of partial opening in adjacent of barrier. Comparison of the flow depths from numerical outcomes and measured experiments showed the numerical scheme of Roe is a robust and capable method for simulation of complicated stationary shocks in shallow water flows. Finally some outcomes such as maximum flow depth in produced shocks and reflection positions have been studied in the dimensionless form.
**KEYWORDS:** Shallow Flows, Numerical schemes, Reimann Solvers, Roe Scheme, Stationary Shocks.

## 1 INTRODUCTION

When occurs an abrupt change in flow depth and/or velocity, the shocks or bores will be produced. The shocks normally appear in supercritical flows while transitory from super to sub-critical regime, named trans-critical, can produce a shock (i.e. hydraulic jump). The governing equations on shock simulation problem are the shallow flow equations. According to their nonlinearity, the analytical solution is limited to simplified cases such as neglecting of some important terms; hence the numerical methods perform as valuable tools to solve the nonlinear shallow flow equations although on the other hand, due to sudden changes of flow pattern by shock adjacent, the simulation of their locations and their shapes are more complicated. Therefore, the conventional numerical methods such as Preissmann (or 4-point scheme) is not capable to capture shocks [1].

Recently, the shock capturing methods are more applicable to shallow flow equations. In 1985 Fenema and Chaudhry applied Beam-Warming, Gabutti and Mc Cormack schemes including artificial viscosity for the simulation of dam-break problem [2]. In start of 1980's decade, the idea of Godunov conservative method has been applied for the solution of Euler equation. These methods are not capable to capture sharp discontinuities such as shock and bores without applying numerical viscosity or simplifications of governing equations [1]. Roe (1981), Van Leer (1977, 1982) and Osher & Solomon (1982) developed the approximate Reimann Solvers in Euler equation [2], [3]. To eliminate non-physical spurious oscillations, Harten (1983) suggests Total Variation Diminishing (TVD) methods [4]. Sweby (1984) and Yee (1987) develop these methods later and in 1990's decade, they were applied by many researchers for the solution of shallow flows. Galister (1988) applied the Roe scheme in shallow flow equations. Alcrudo and Garcia Navarro (1993), applied the Godunov-type finite volume method for the solution of shallow flow equations [5]. Zhao et al. (1996) used the approximate Reimann solvers of Osher & Solomon for the simulation of unsteady flow in rivers [6]. Chipada et al. (1998) used the Roe scheme to modeling of flow through transitions and tidal waves. Also Yang and Greimann (2000), applied TVD version of Roe scheme for the simulation of dam-break problem with sediment transport [6]. Tseng (2001) used high-resolution conservative methods for the simulation of wave in channels [7]. Callefi et al. (2003) applied approximate Reimann solver of HLL for the simulation of extreme floods in rivers [8]. Ghezelsofloo and Jaefarzadeh (2005) applied the TVD version of Roe scheme to simulation of stationary oblique shocks which are appear as a result of confrontation of a supercritical flow to cross-barrier in the open channel flow [9]. Sheng Bi et al. (2015) applied improved unstructured finite volume algorithm in 2D shallow flow modeling [10]. Martines et al. (2017) offers a detailed validation of finite volume (FV) flood models in the case where horizontal floodplain flow is affected by sewer surcharge flow via a manhole [11].

The positions oblique and complicated stationary shocks generating by partial opening in a super-critical is complicated. The lateral component of the velocity makes the flow hit to channel wall and it generates the oblique shocks that developed successively and decrease the sharpness and height to the downstream. The conventional numerical schemes are not able to capture

*Corresponding Author: Abbas Ali Ghezelsofloo, Assistant Professor, Civil Engineering Department, Islamic Azad University, Mashhad Branch, Iran. Email:ghezelsofloo@mshdiau.ac.ir

position and height of stationary shocks. In this paper the developed shocks in the mentioned case have been simulated in two-dimension with Roe-TVD numerical scheme. The numerical results will be calibrated with some experimental data from a laboratorial setup. Then we have studied dimensionless behavior of produced shocks with various percentages of partial openings. Subsequently, for calibration of the model, we used some laboratorial experiments.

## 2 THE GOVERNING EQUATIONS

The governing equation for the two-dimensional modeling of shocks and bores is the shallow water equations which are derived by the depth averaging of the three-dimensional Navier-Stokes equations by considering a hydrostatic pressure distribution and incompressible flow. The governing equations, based on the conservation of mass and momentum, for the two-dimensional unsteady flow in a rectangular channel, may be derived as:

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

In equation (1), $\boldsymbol{U}$ is the vector of the conserved variables, $\boldsymbol{F(U)}$ and $\boldsymbol{G(U)}$ are the flux vectors in the *x*- and *y*- directions, respectively, and $\boldsymbol{S(U)}$ is the vector of source terms with the considering only the effects of bottom slope and friction in the x- and y- directions may be written as equation (4).

$$U=\begin{bmatrix} \mathrm{h} \\ \mathrm{hu} \\ \mathrm{hv} \end{bmatrix} \qquad (2)$$

$$\mathrm{F(U)}=\begin{bmatrix} hu \\ hu^2+0.5gh^2 \\ huv \end{bmatrix} \; ; \; \boldsymbol{G(U)}=\begin{bmatrix} hv \\ huv \\ hv^2+0.5gh^2 \end{bmatrix} \qquad (3)$$

$$S(U)=\begin{bmatrix} 0 \\ \mathrm{gh}(\mathrm{s_{ox}}-\mathrm{s_{fx}}) \\ \mathrm{gh}(\mathrm{s_{oy}}-\mathrm{s_{fy}}) \end{bmatrix} \qquad (4)$$

In Eq. (4), $s_{ox}$ ($s_{oy}$) is the bottom slope and $s_{fx}$ ($s_{fy}$) stands for friction slope in the *x*- *(y-)* direction, respectively, which may be estimated from the Manning equation.

## 3 NUMERICAL MODELING

The differential form of conservation laws is only able to simulate the flow in smooth regions. In discontinuous regions, however, an integral form of governing shallow flow equation may be applied to including the weak solutions as well. Integration of equation (1) for a computational cell (i) in the x-t plane leads to a general conservative formula which with considering the source terms, it may be written as:

$$\boldsymbol{U}_i^{n+1}=\boldsymbol{U}_i^n-\frac{\Delta t}{\Delta x}\left[\boldsymbol{F}_{i+1/2}^n-\boldsymbol{F}_{i-1/2}^n\right]+\Delta t\,\boldsymbol{S}_i^n \qquad (5)$$

Where superscript *n* denotes for computing time level, $\Delta t$ is the time step, and $\Delta x$ is the cell width. $\boldsymbol{F}_{i-1/2}$ and $\boldsymbol{F}_{i+1/2}$ stand for input and output numerical fluxes to the computational cell (*i*), respectively.

The approximate Reimann solvers on finite volume frameworks are known also as 'high-resolution finite volume methods'. In FDS methods, since the shallow flow equations system is from parabolic type and includes the inherent character of signal propagation, numerical fluxes are derived on the base of local information of wave propagation. To this extent, they have special capability for capturing the sharp discontinuities. The FDS methods are derived on the base of Godunov method, therefore they are known as 'Godunov-type' methods. The numerical fluxes in Roe scheme have been derived explicitly on the base of upwinding technique (Glaister, 1988).

The boundary conditions are divided into two general categories of open and solid boundaries. For current simulation in the longitudinal channel walls we have solid boundary as well as upward and downward of barrier.

## 4 APPLICATION OF THE MODEL

The presence of a cross-barrier partially in width of an open channel make increase of water depth in upstream (backwatering), then according to the flow releasing downstream beside of contraction and the increasing the velocity and make it as a supercritical flow. The lateral component of the velocity makes the flow hit to channel wall and it generates the oblique shocks that developed successively and decrease the sharpness and height to the downstream. Figure 1 shows the 3D view of cross-barrier in an open channel. Following, the developed shocks in the mentioned case have been simulated in two-dimension with Roe-TVD numerical scheme. The numerical results will be compared with some experimental data from a laboratorial setup.

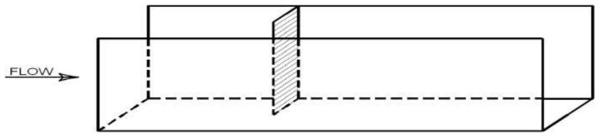


**Figure 1. The 3D view of a flume with a cross-barrier**

The modeling reach has a longitudinal length of L=4 m that cross-barrier is located at Lu=1 m from upstream of reach. Due to importance of produced shocks at downstream, the simulation domain includes 1 m in upstream of barrier and 3 m in downstream. The flume width is B=0.4 m with bed slope of $S_0$=0.006625. The barrier lengths are in two cases $L_b$=0.12 m and $L_b$=0.16 m that are located in normal direction of flow, partially. The numerical modeling grid size is $\Delta x = \Delta y = 0.02m$. Figures 3 shows the 3D view and contour lines of flow depths resulted from numerical simulation with barrier length 0.12 m. In general, it can be seen that numerical modeling is able to simulate shock pattern in comparison with the experimental observation shown in figure 2.

**Figure 2. General view of the experimental setup and developed shocks**

## 5. LABORATORIAL SETUP AND MEASUREMENTS

In the present research, a laboratorial flume has been used which located in the hydraulic lab of Ferdowsi university of Mashhad (Iran). The general view of the laboratorial setup is shown in Figure 2. The material of flume's wall is the Perspex and the bed is steel that manning roughness is estimated n=0.01 by some laboratorial tests. The steel barrier with 3 mm thickness is located on left channel wall, normally in flow direction (figure 1) and sealed well. When the pump is start, after few seconds, the flow got stationary state and one can observe successive stationary and oblique shocks that appear in downstream of barrier.

The water depths have been measured in 3 longitudinal sections and 12 cross-sections which are shown in figure 7. The measured flow depths data in longitudinal profiles are with increments of $\Delta X$ =4 cm and latitudinal profiles with increments of $\Delta Y$ =2 cm. we have used a scale with 1 mm error accuracy.

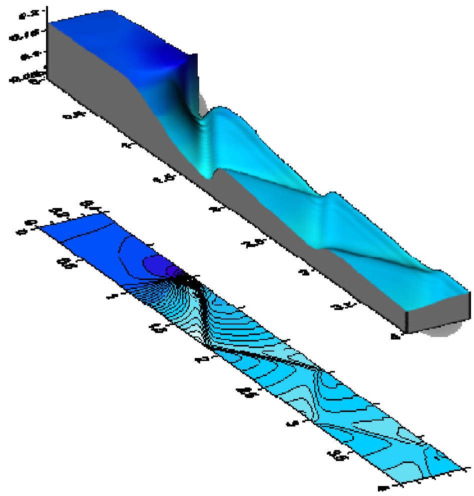

**Figure 3. The 3D view and counter lines of depths resulted from numerical scheme**

Some divergences in numerical results in comparison with the experimental flow depth data may be resulted from:

1) Relative error in measuring of the depth and the flow discharge which is due to the extreme oscillation and turbulence of the flow, especially in the first downstream shock that includes a barrier.
2) Despite of the fact that the actual thickness of the barrier is about 3mm, in the numerical modeling it is considered equal to 0.02m.
3) Although the flow condition is 3D around the barrier, 2D shallow water equations (averaged depth) are applied in the numerical model.
4) The applied numerical model includes a second order accuracy. Although the previous experiments and also the latest results of this modeling shows the capability of this numerical method to capture the shocks, applying more accurate equations will be more helpful.
5) The first step which is created in the first downstream reflection is produced from the extreme jet flow which is faced to a wall. It should be mentioned that the present theoretical model is not able to simulate such flows.

6 MODEL CALIBRATIONS

Since the theoretical model fails to consider all genuine physical conditions, the calibration model should be done to fit numerical outcomes and experimental data. In this simulation, the effects of turbulence head losses are not taken into account, therefore the turbulence modeling should be combined to the selected numerical scheme. A simple method that may be used to calibrate the numerical model is to take into account the above said effects in the source terms. As already indicated, in numerical modeling the head loss due to effects of turbulence is not included in the source terms vector. Consequently, one can consider them along with physical roughness in the manning roughness. Therefore the total manning coefficient will be:

$$n_t = n + n' \qquad (6)$$

In this equation, n is Manning roughness coefficient in ordinary condition, $n'$ is roughness equal to turbulence head loss and $n_t$ is total Manning roughness considered in the downstream of the barrier (shocks developing area). Following, the problem has been simulated for two different cases of barrier length of 12 and 16 cm, respectively.

For the case of $L_b$ = 12 cm, the problem has been solved for the various amount of $n'$, finding the best fitness between numerical and experimental results at $n'$=0.004. The three longitudinal profiles for $n'$ equal to 0.00, 0.003 and 0.005 are shown in figures 4 to 6.

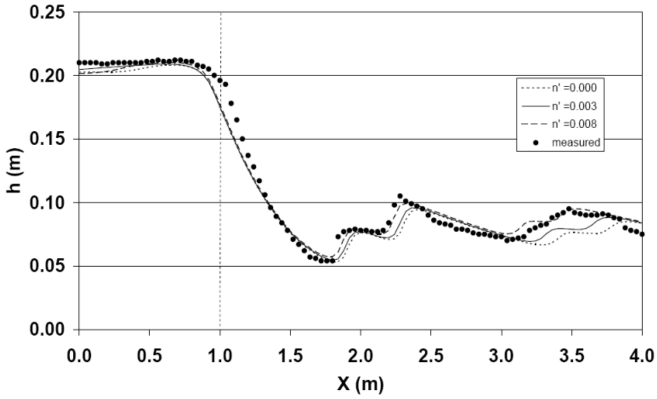


**Figure 4. The longitudinal profile with various $n'$ for P1 (10 cm from channel left wall) for $L_b$=12 cm**

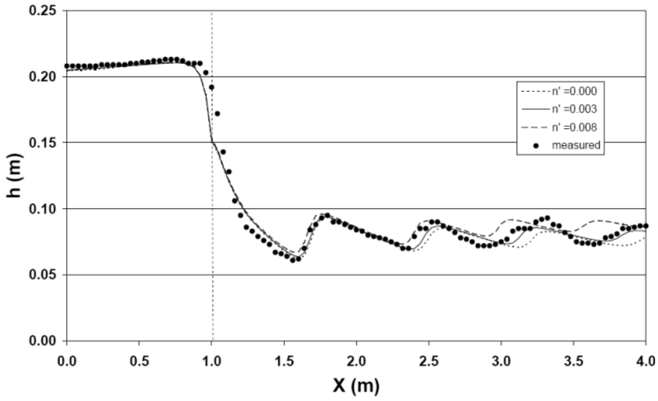


**Figure 5. The longitudinal profile with various $n'$ for P2 (20 cm from channel left wall) for Lb=12 cm**

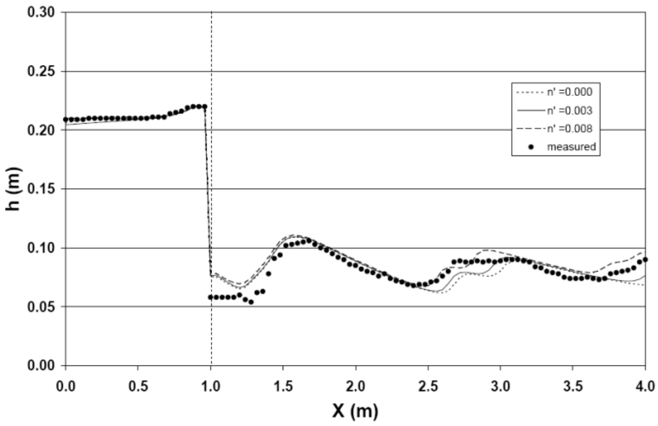


**Figure 6. The longitudinal profile with various $n'$ for P3 (30 cm from channel left wall) for $L_b$=12 cm**

As for the case of $L_b$ = 16 cm, the laboratorial experiments suggest that as the barrier length increases, the turbulence is subject to increases as well with sharper shocks and more height. Therefore we look forward to more amounts for $n'$. In this case for the various amount of $n'$ from 0.00 to 0.011 (with increment of 0.001) the problem was simulated and we found the best fitness between numerical and experimental results at $n'$=0.008. The longitudinal profiles for various $n'$ were shown in figures 7 to 9.

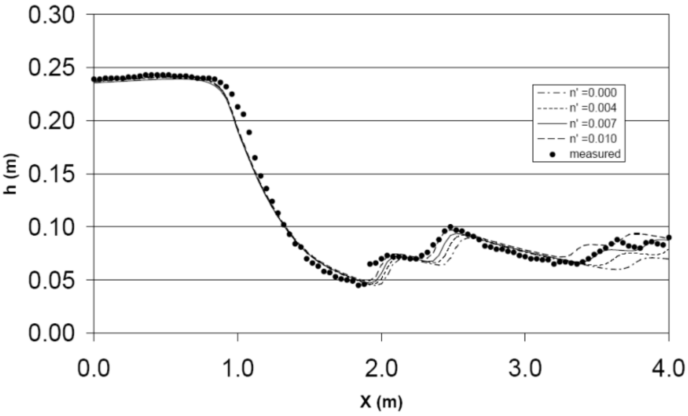


**Figure 7. The longitudinal profile with various $n'$ for P1 (10 cm from channel left wall) for $L_b$=16 cm**

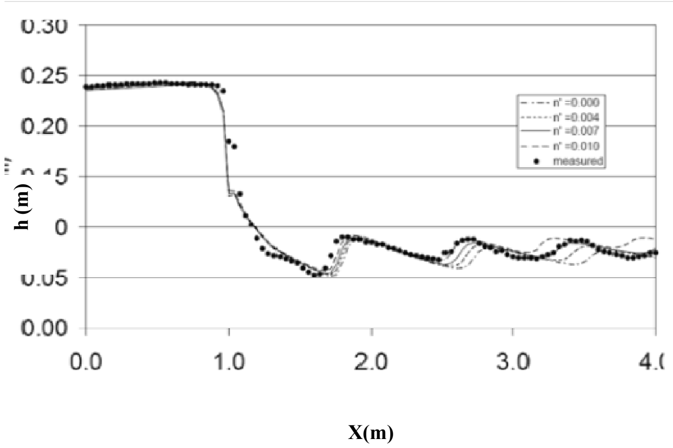


**Figure 8. The longitudinal profile with various $n'$ for P2 (20 cm from channel left wall) for Lb=16 cm**

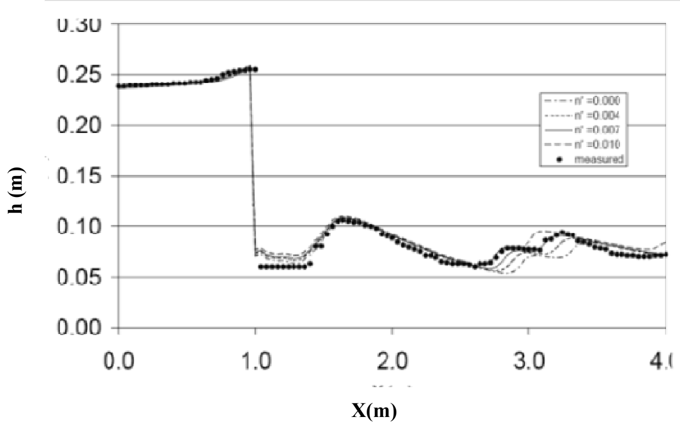


**Figure 9. The longitudinal profile with various $n'$ in P3 (30 cm from channel left wall) for Lb=16 cm**

**7 EVALUATION OF SHOCK CHARACRTISTICS AGAINST BED SLOPE AND ROUGHNESS CHANGE**

The downstream shocks (height and position) sensitivity is determined against channel roughness in the numerical model calibration. This section investigates behavior of the produced shocks in various bed slopes. With this regard the problem has been simulated bed slopes of $S_0$=0.006625 (slope of experiments), $S_0$=0.01, $S_0$ =0.02 and $S_0$=0.03 in barrier length of $L_b$=12 and 16 cm.

The numerical results show that, when the channel slope increases, the shocks get farther from the barrier. Consequently, a change in shock characteristics against slope changes and channel roughness is illustrated while applying numerical model.

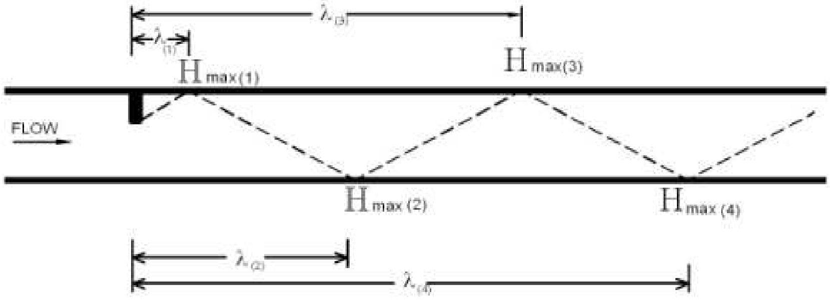


**Figure 10. Definition of shock characters at downstream of the barrier**

This figure illustrates s the maximum shock heights and the distance of shock encounter with wall shown by $H_{max(i)}$ ,$\lambda_{(i)}$, respectively. As already mentioned, in the upstream boundary condition the height of water ($H_u$) is constant. In consideration of the bed slope and roughness, one can use normal depth ($H_0$) from Manning equation. Consequently the dimensionless parameter of $H_u / H_0$ is defined. Accordingly shock characteristics in dimensionless form of $H_{max(i)} / H_0$ and $\lambda_i / B_i$ are determined and plotted in figures 11 to 14.

As the figures above indicate the downstream shocks are developed following a similar pattern in barriers with various lengths. The first shock reflection position landa1 is similar to all cases. However the next reflection positions increase gradually specially in the case of downstream shocks. Meanwhile the maximum shock heights in consequent shocks decreases with $H_u / H_0$ reaching a normal depth.

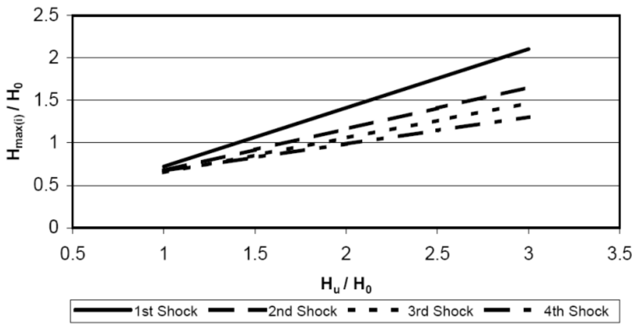


**Figure 11. The dimensionless characteristics for maximum shock height for Lb=12 cm**

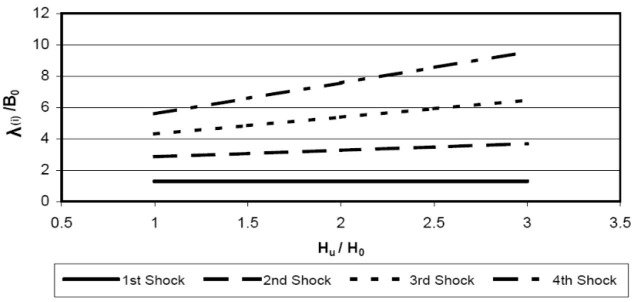


**Figure 12. The dimensionless characteristics for distance shock encounter for Lb=12 cm**

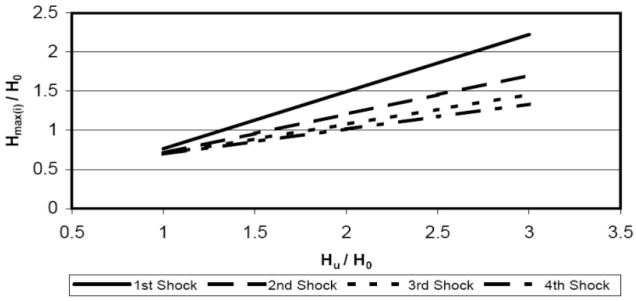


**Figure 13. The dimensionless characteristics for maximum shock height for $L_b$=16 cm**

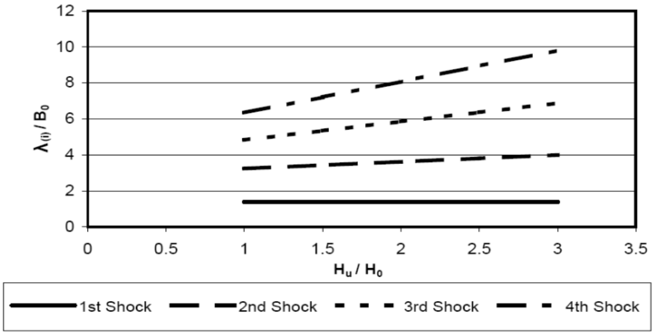


**Figure 14. The dimensionless characteristics for distance shock encounter for Lb=16 cm**

## 8 CONCLUSION

The conservative methods based on Reimann solvers are able to capture discontinuities and shocks properly. In the present paper, an approximate Reimann solver of Roe (TVD-version) has been employed to simulate the stationary shocks in an open channel. The numerical modeling in general, simulates the developed shocks behaviors. An experimental setup along with measurements showed the need for calibration. Applying a simple method, manning roughness coefficient was added to consider the turbulence effects. Following, the shock characteristics including maximum height and distance shock encounter were plotted dimensionless. The first shock which is created in the primary reflection of downstream is produced from the extreme jet flow which is faced to a wall. It should be mentioned that the present theoretical model is not able to simulate such flows. In general, comparison of the flow depths from numerical outcomes and measured experiments showed the numerical scheme of Roe is a robust and capable method for simulation of complicated stationary shocks in shallow water flows. Finally some outcomes such as maximum flow depth in produced shocks and reflection positions have been studied in the dimensionless form which may useful for study of hydraulic structures and drainage network systems.

## 9 REFRENCES

[1]. Toro E. F. (2001), "Shock–capturing methods for free–surface shallow flows", John Wiley & Sons LTD.

[2]. Roe P.L. (1981), "Approximate Riemann solvers, parameter vectors, and difference Schemes", Journal of Computational Physics, 43, pp. 357-372.

[3]. Osher S. (1984), "Riemann solvers, the entropy condition, and difference approximation", SIAM Journal of Numerical Anal. Vol. 21, No. 2.

[4]. Hartern, A. (1983), "High resolution schemes foe conservation laws", Journal of Computational Physics No. 49.

[5]. Alcrudo F. And Garcia-Navarro P. (1993), "A high resolution Godunov-type scheme in finite volumes for the 2D shallow–water equations", International Journal for Numerical Methods in Fluids, Vol. 16, pp. 489-505.

[6]. Zhao D. H., Shen H. W., Lai J. S. and Tabios III G. Q. (1996), "Approximate Riemann solvers in FVM for 2D hydraulic shock wave modeling", Journal of Hydraulic Engineering, Vol. 122, No. 12., pp. 692-702.

[7]. Tseng M. H., Hsu C.A. and Chu C. R. (2001), "Channel routing in open–channel flows with surges", Journal of Hydraulic Engineering, Vol 127, No. 2, pp. 115-122.

[8]. Calleffi V., Valiani A. and Zanni A. (2003) "Finite volume method for simulating extreme flood events in natural channels", Journal of Hydraulic Research, Vol. 41. No. 2, pp. 167-177.

[9]. Ghezelsofloo A. A. and Jaefarzadeh M. R. (2005), “Numerical modeling of dam-break problem with a high resolution finite difference method", International Journal of Engineering Science, Vol. 16, No. 4

[10]. Sheng Bi, Lixiang Song, Jianzhong Zhou, Linghang Xing, Guobing Huang1, Minghai Huang and Suncana Kursan (2015), "Two-dimensional shallow water flow modeling based on an improved unstructured finite volume algorithm", Advances in Mechanical Engineering, Vol. 7(8) 1–13

[11]. Martins R., Kesserwani G.,Rubinato M., Lee S., Leandro J., Djordjević S. and Shucksmith J. D. (2017), "Validation of 2D shock capturing flood models around a surcharging manhole", Urban Water Journal, Volume 14, Issue 9