Experimental Analysis and Numerical Simulation of Hydraulic Jump

B Nandi¹, S Das² and A Mazumdar³

1 PhD Student, School of Water Resources Eng., Jadavpur University, Kolkata, India
2 Assistant Professor, School of Water Resources Engineering, Jadavpur University, Kolkata, India
3 Professor & Director, School of Water Resources Engineering, Jadavpur University, Kolkata, India
E-mail: subhasishju@gmail.com

Abstract. Saint Venant equations are numerically solved to simulate the formation of hydraulic jump in a rectangular channel having a small bed slope. The MacCormack’s scheme is used for the solution by applying specified initial and boundary conditions until a steady state flow is reached. The location of the hydraulic jump is determined as a part of these computations. The artificial viscosity technique should be used in the computations to dampen the superior oscillations near the steep gradient of the simulated hydraulic jump. Twenty laboratory experiments were carried out for verification of the numerical model. The upstream Froude number for these experiments ranged from 2.17 to 7.0 in three different bed slopes 0, 0.02174, 0.0475. The simulated hydraulic jump profiles using the MacCormack’s scheme shows a good agreement with the experimental data. An empirical equation was developed to determine the location of hydraulic jump using regression analysis based on simulated data. Software based on Computational Fluid Dynamics (CFD) was also used to simulate two of these experiments. The results obtained from CFD analysis matched fairly with the experimental results.

1. Introduction
Hydraulic jump is a sudden transition to subcritical flow from a supercritical flow. The phenomenon is observed in canals below sluice gates, at the foot of spillways or when there is an abrupt change in slope from steep to flat. Hydraulic jumps are practically used as energy dissipaters below spillways to avoid scouring in the downstream, for mixing chemicals and aerate water for city water supplies or for removing air pockets from water supply lines for preventing air locking. It is essential to determine the various parameters like length and location of the jump, amount of energy dissipated etc. to design a hydraulic structure.

The theory of jump was developed in earlier days mainly on the basis of extensive experimental and empirical work. Several researches were done by conducting extensive laboratory experiments and developing design charts and empirical relationships between various flow parameters [1-11]. The advent of high speed digital computers and developments in Computational fluid dynamics gave the opportunity to solve the governing equations of hydraulic jump numerically. An extensive amount of data has been reported in the literature on the hydraulic jump. To determine the jump location, computed the water surface profiles were computed for supercritical flow starting from the upstream end and the subcritical flow starting from the downstream end, and the jump is formed at a location...
where the specific forces on both sides of the jump are equal \[12\]. A strip-integral method was used to compute the jump length, water surface profile, and pressures at the bed \[13\]. The finite-difference method was used \[14\] and the finite-element method was used \[15\] to solve the St. Venant equations numerically until a steady state was reached. The location of the hydraulic jump is automatically computed as part of the solution. Boussinesq equations describing one-dimensional unsteady, rapidly varied flows were integrated numerically to simulate both the sub- and supercritical flows and the formation of a hydraulic jump in a rectangular channel having a small bottom slope. For this purpose the MacCormack (second-order accurate in space and time) and two-four (second-order accurate in time and fourth-order in space) explicit finite-difference schemes were used to solve the governing equations subject to specified end conditions until a steady state was reached. There was significant agreement between the experimental and numerical results. The fourth order accurate model was found to be more precise than the second order accurate model. But the simulations showed that the Boussinesq terms have little effect in determining the location of the hydraulic jump. Hydraulic jump was numerically simulated on a straight horizontal channel with supercritical Froude numbers 2.0 and 4.0 by solving Reynolds averaged Navier-Stokes equations \[16\].

Turbulence was modeled through the \( k - \varepsilon \) closure equations. Galerkin finite element method with three-noded triangular elements is used for spatial discretization. A detailed study of the internal and external characteristics of hydraulic jump is done and compared with experimental values where possible. The FLOW 3D was used to simulate the hydraulic jump in the convergence stilling basin \[17\]. The software was applied to numerically solve the Navier–Stokes equations for solution domains, namely the shout, the stilling basin and the downstream of dam, and to estimate the turbulence flow, the standard k-\( \varepsilon \) and RNG models was used. These models are based on the volume-of-fluid method, and they are capable of simulating the hydraulic jump. The calculated results such as the pressure, the velocities, the flow rate, the surface height air entranced, the kinetics energy, the kinetics energy dissipated, and the Froude number were compared with the scale model data where available. The physical model and CFD model results showed good correlations. The primary goal of this present study is to numerically simulate hydraulic jump in a sloping channel using the MacCormack method \[18\] to solve the St. Venant equation which is used as the governing equations. A source code is written in MATLAB (matrix laboratory), which is a proprietary product of MathWorks, to do the numerical computations.

2. Governing equations

Unsteady flow phenomenon such as hydraulic jump in rectangular channel are often modeled as one dimensional flow which can be described by a set of quasi-linear, hyperbolic partial differential equations, called the Saint-Venant equations. The derivation of the equations involves the following assumptions:

a) The pressure distribution is hydrostatic.
b) The channel bottom slope is small enough such that the depth measured normal to the channel bed is approximately equal to the depth measured vertically.
c) The velocity distribution is uniform over the entire channel cross-section.
d) The channel is prismatic.
e) Friction losses for a given flow velocity during unsteady flow is same as that during steady flow.

The one dimensional St. Venant equation may be written as follows:

\[
\begin{align*}
\text{Continuity:} & \quad \frac{\partial h}{\partial t} + \frac{\partial (uh)}{\partial x} = 0 \\
\text{Momentum:} & \quad \frac{\partial (uh)}{\partial t} + \frac{\partial }{\partial x} \left( hu^2 + \frac{1}{2} gh^2 \right) = gh \left( S_o - S_f \right)
\end{align*}
\]

where; \( x \) is the distance along the channel bottom (considered positive in the downstream direction), \( t \) is the time, \( u \) is the flow velocity in the \( x \)-direction, \( h \) is the flow depth, \( g \) is the acceleration due to gravity, \( S_o \) is the channel bottom slope, and \( S_f \) is the slope of the energy grade line. Manning equation (1889) may be used to calculate the friction slope.
3. Numerical scheme

A hydraulic jump can be simulated by solving the above governing equations with appropriate boundary conditions. The initial conditions are specified and the iterations are continued until steady state is reached. If shock capturing numerical technique is used, then the jump forms as a part of the steady state solution. MacCormack’s scheme [18] is a two level predictor corrector scheme. In the predictor part, forward finite difference is used for the spatial derivative terms and backward finite difference approximation is used in the corrector part. The finite difference expressions are:

**Predictor step:**

\[
\frac{\partial f}{\partial t} = \frac{f^*_i - f^k_i}{\Delta t}
\]

or,

\[
\frac{\partial f}{\partial x} = \frac{f^*_i - f^k_i}{\Delta x}
\]

where \(i\) is the node in \(x\) direction, \(k\) is the node in time direction, asterisks refers to the predicted values of the variable; and \(\Delta x\) and \(\Delta t\) are the spatial grid size and time step size respectively. Based on the above finite difference approximations (1) and (2) may be written for determining the predicted values \(h^*_i\) and \(u^*_i\):  

\[
h^*_i = h^k_i - \frac{\Delta t}{\Delta x} \left( u^k_i h^k_i - u^k_i h^k_{i+1} \right)
\]

\[
u^*_i h^*_i = u^k_i h^k_i - \frac{\Delta t}{\Delta x} \left\{ \left( u^k_{i+1} \right)^2 h^k_{i+1} + \frac{g}{2} \left( h^k_{i+1} \right)^2 \right\} - \left\{ \left( u^k_i \right)^2 h^k_i + \frac{g}{2} \left( h^k_i \right)^2 \right\} \left( S_0 - S_f \right)
\]

Two double asterisks denote the values of the variable after the corrector step. Based on the above finite difference approximations the variables after the corrector step can be written as:

\[
h^{**}_i = h^*_i - \frac{\Delta t}{\Delta x} \left( u^*_i h^*_i - u^*_i h^*_{i+1} \right)
\]

\[
u^{**}_i h^{**}_i = u^*_i h^*_i - \frac{\Delta t}{\Delta x} \left\{ \left( u^*_i \right)^2 h^*_i + \frac{g}{2} \left( h^*_i \right)^2 \right\} - \left\{ \left( u^*_i \right)^2 h^*_{i+1} + \frac{g}{2} \left( h^*_{i+1} \right)^2 \right\} \left( S_0 - S_f \right)
\]

The channel is divided into \(n\) equal reaches. Thus if the upstream end is numbered section 1, then the downstream end will be \(n+1\). The flow velocity and flow depth is specified at time \(t=0\) as the initial condition. The flow is assumed to be supercritical initially in the entire channel. The initial steady state flow depth is determined by integrating the gradually varied flow equation; starting with the specified flow depth and velocity at section 1:

\[
\frac{dh}{dx} = \left( S_0 - S_f \right) \left( 1 - \frac{u^2}{gh} \right)^{-1}
\]

The MacCormack’s scheme is used to compute the variables at the interior nodes. At the boundaries the flow conditions are specified. The flow depth and velocity are specified at the upstream boundary and they are same as the initial conditions. At the downstream boundary, a constant flow depth is specified and the flow velocity is calculated from the characteristic form of (1) and (2) using a forward
finite-difference approximation. A modified-equation analysis of finite-difference schemes [19] shows that higher order terms are introduced which are not present in the governing partial differential equations. These terms represent the truncation errors which affect the behavior of the scheme and contribute to the scheme what is known as diffusion and dispersion. The solution has dissipative errors if the leading term in the truncation error contains an even derivative and dispersive errors if the leading term has odd derivatives [20]. For $C_n$ less than that required by the Courant-Friedrich-Lewy (CFL) limit, the dispersive errors result in introducing numerical oscillations in the solution. Therefore it becomes necessary to add artificial viscosity to smooth these oscillations. Several procedures have been reported for this purpose.

A procedure given in [21] used herein, has the advantage of smoothing regions where the solution has large gradients while leaving relatively smooth areas undisturbed; i.e., high-frequency oscillations are smoothed. A parameter $\Xi$ is first computed from a normalized form of the gradients of one variable. For the studies reported herein, the depth of flow $h$ was selected for determining the parameter:

$$\Xi_j = \frac{\Delta x}{\Delta t} \left| \frac{h_{j+1} - 2h_j + h_{j-1}}{h_{j+1}} \right| + 2\left| h_j \right| + \left| h_{j-1} \right|$$

$$\Xi_j^{1/2} = k \frac{\Delta x}{\Delta t} \max\left(\Xi_{j+1}, \Xi_j\right)$$

In which $k$ is used to regulate the amount of dissipation. The computed variables are then modified as

$$f_i^{k+1} = f_i^{k+1} + \Xi_i^{1/2} \left( f_{i+1}^{k+1} - f_i^{k+1} \right) - \Xi_i^{1/2} \left( f_i^{k+1} - f_{i-1}^{k+1} \right)$$

4. Experimental details

The experiments were carried out in a rectangular perspex flume 36.5 cm width, 45 cm height and 5 m long (Figure 1). The discharge was measured by a digital (magnetic type) flow meter. The flume has a tail gate to control the water depth. By adjusting the tail water depth the jump position was varied. The flow depths at the upstream end and in the section of the flume with metal walls were measured at equally spaced intervals by a point gauge having an accuracy of 1 mm. There were continuous undulations in the water surface downstream of the jump. The maximum and minimum levels of these undulations at a location were marked and an average of these levels was considered the depth at that location. Twenty laboratory experiments were carried out with the Froude number upstream of the jump ranging from 2.17-7.00 for three different slopes 0.0435, 0.02174 and 0.

5. Numerical model

A second order accurate numerical model is developed. The St. Venant equations are solved by the MacCormack’s scheme [18]. The time-step size was restricted by the Courant stability condition and the spatial grid size. The Courant number was set equal to 0.65 since best results are obtained when it is approximately equal to 2/3. To smooth the high-frequency oscillations near the jump, the dissipation coefficient $k$ in the Jameson’s formula, was determined by a trial-and error procedure. It is desirable to keep its value as low as possible while still smoothing the high-frequency oscillations. Trials with values ranging from 0.01 to 0.05 indicated that a value of 0.03 provided the best results. To run the numerical model, the flow depth and velocity at the upstream end and only the flow depth at the downstream end are specified. The upstream flow velocity for each run was computed from the continuity equation $Q = Bhu$ where $Q$ is discharge and $B$ is flume width, $h$ is measured depth and $u$ is flow velocity. The Froude number of the incoming flow is determined from the equation $Fr = u/(gh)^{0.5}$. The upstream flow depth, velocity, Froude numbers, and downstream flow depth for different runs are listed in Table 1. In these runs, the Manning coefficient ($M_n$) for the flume was determined by trial and error by matching the computed water surface profile with the measured water levels in the flume during the initial steady supercritical flow. For the range of Froude numbers tested, these values varied from 0.014 to 0.016 depending upon the flow depth since the bottom of flume is made up of metal and
the sides are made up of glass. First, the initial steady-state depth and velocity at every computational node were computed by assuming the flow to be supercritical throughout the flume. Then, the unsteady computations were started by increasing the downstream depth to the value measured during the experiment (see Table 1). The computations were continued until they converged to the final steady state for the specified end conditions.

One very important parameter in the simulation of a hydraulic jump is the size of the spatial grid size, $\Delta x$. Its value was varied from 0.1 m to 0.4 m. Values greater than 0.4 m and in some cases 0.3 m could not be used since they resulted in the jump forming at less than three computational nodes. For $Fr = 7$, and slope 0.0435 the jump was computed to form between 1.60 m and 2.10 m from the upstream end of the flume, with the average distance of four different values of $\Delta x$ being 1.95 m. In other words, the jump location may be predicted in a satisfactory manner for typical engineering applications by using a reasonable value of $\Delta x$.

![Figure 1. Experimental pictures showing hydraulic jump.](image)

For the range of Froude numbers tested, these values varied from 0.014 to 0.016 depending upon the flow depth since the bottom of flume is made up of metal and the sides are made up of glass. First, the initial steady-state depth and velocity at every computational node were computed by assuming the flow to be supercritical throughout the flume. Then, the unsteady computations were started by increasing the downstream depth to the value measured during the experiment (Refer Table 1). The computations were continued until they converged to the final steady state for the specified end conditions. One very important parameter in the simulation of a hydraulic jump is the size of the spatial grid $\Delta x$. Its value was varied from 0.1 m to 0.4 m. Values greater than 0.4 m and in some cases 0.3 m could not be used since they resulted in the jump forming at less than three computational nodes. For $Fr = 7$, and slope 0.0435 the jump was computed to form between 1.60 m and 2.10 m from the upstream end of the flume, with the average distance of four different values of $\Delta x$ being 1.95 m. In other words, the jump location may be predicted in a satisfactory manner for typical engineering applications by using a reasonable value of $\Delta x$. 

5
Table 1. Experimental details

| Experiment Number | Discharge (m³/s) | Upstream Depth (m) | Upstream Froude Number | Bed Slope | Downstream Depth (m) |
|-------------------|------------------|--------------------|-------------------------|-----------|----------------------|
| A 1               | 19.1             | 0.039              | 2.30                    | 0.04350   | 0.234                |
| A 2               | 19.1             | 0.033              | 2.90                    | 0.04350   | 0.200                |
| A 3               | 19.1             | 0.040              | 2.17                    | 0.04350   | 0.260                |
| A 4               | 19.1             | 0.032              | 3.06                    | 0.04350   | 0.268                |
| A 5               | 19.1             | 0.027              | 3.83                    | 0.04350   | 0.215                |
| A 6               | 19.1             | 0.026              | 4.23                    | 0.04350   | 0.265                |
| A 7               | 19.1             | 0.023              | 5.00                    | 0.04350   | 0.272                |
| A 8               | 19.1             | 0.020              | 6.18                    | 0.04350   | 0.273                |
| A 9               | 19.1             | 0.018              | 7.00                    | 0.04350   | 0.285                |
| A 10              | 19.1             | 0.039              | 2.30                    | 0.02174   | 0.167                |
| A 11              | 19.1             | 0.033              | 2.90                    | 0.02174   | 0.170                |
| A 12              | 19.1             | 0.040              | 2.17                    | 0.02174   | 0.182                |
| A 13              | 19.1             | 0.032              | 3.06                    | 0.02174   | 0.160                |
| A 14              | 19.1             | 0.027              | 3.83                    | 0.02174   | 0.175                |
| A 15              | 19.1             | 0.026              | 4.23                    | 0.02174   | 0.205                |
| A 16              | 19.1             | 0.023              | 5.00                    | 0.02174   | 0.215                |
| A 17              | 19.1             | 0.020              | 6.18                    | 0.02174   | 0.205                |
| A 18              | 11.2             | 0.023              | 3.29                    | 0         | 0.070                |
| A 19              | 15.0             | 0.027              | 3.06                    | 0         | 0.080                |
| A 20              | 19.1             | 0.027              | 3.83                    | 0         | 0.110                |

6. Comparative analysis between experimental and numerical results

Once the numerical solution converged to a steady state - the depths at the corresponding grid points is obtained, which gives the flow profile alongwith the jump. Figures 2-3 give a comparative view between numerical and experimental results for flows with different pre jump Froude number at different bed slopes. The depth of flow in meters is plotted in the vertical axis and distance along the flume, plotted in the horizontal axis. The profile obtained from experiment is plotted in continuous line and numerical profile is plotted in dashed line. The St. Venant equation assumes hydrostatic pressure distribution. But the pressure distribution in rapidly varied flow phenomenon like hydraulic jump which has steep water surface gradients is not hydrostatic.

![Figure 2. Comparative analysis between numerical and experimental results.](image)
Equations describing these flows were derived in [22] and [23] assuming that the vertical velocity varies from zero at the channel bottom to its maximum value at the free surface. These equations referred to as the Boussinesq equations. But the Boussinesq terms have little effect in predicting the jump location [24]. Numerical models gave the same result for the St. Venant equation and Boussinesq equations when fourth order accurate numerical scheme is used. Thus the accuracy was more dependent on the order of the numerical scheme. MacCormack scheme being second order accurate in space and time could not predict the jump location precisely for higher Froude numbers but gave good results for low Froude numbers as is evident from the comparison of the results. The comparison between the computed and the numerical results shows that as the Froude number increases the jump forms downstream of the location obtained experimentally. For the slope 0.0435 for Froude numbers more
than 4.23 the jump is located downstream of the location measured experimentally by a distance of 0.25 m to 0.45 m. For the slope 0.02174 the jump location is about 0.26 m to 0.475 m downstream from that measured. The jump location is simulated better at lower Froude numbers and matches closely for Froude numbers 3.83 and less. The overall comparison shows, as shown in figure 4, the deviation of the jump location obtained numerically from the measured location is within ±25%.

7. General equation using regression analysis

Sometimes it may be useful to form an empirical relationship between the parameters determining the jump location. The results obtained from the empirical equation are then compared with the numerical results. It is assumed that the dependent variable \( L \), which is the distance from the beginning of the flume to the location of hydraulic jump is a function of the following independent variables: density of flow \( \rho \), the upstream water depth \( h_u \), the tail water depth \( h_t \) at the end of the channel, the upstream velocity \( u \), the acceleration of gravity \( g \), bed slope \( S_0 \). The general function relationship between the above variables can be written as:

\[
f(L, \rho, h_u, h_t, u, g, S_0) = 0
\]

Using the dimensional analysis, the \( \pi \) terms obtained are, \( \pi_1 = L/h_u \), \( \pi_2 = h_l/h_u \), \( \pi_3 = F_r \) and \( \pi_4 = S_0 \). These \( \pi \) terms may be arranged in the following non dimensional form:

\[
f(L/h_u, h_l/h_u, F_r, S_0) = 0
\]

The general form of equations relating a dependent \( \pi \)-term with a number of independent \( \pi \) terms using regression analysis in this work is in the form of the product of powers of relevant \( \pi \) terms, i.e.,

\[
\pi_1 = C \pi_2^{a_2} \pi_3^{a_3} \cdots \pi_m^{a_m}
\]

Equation 19 can be transformed to a linear expression by taking logarithms of both sides of the equation, as follows:

\[
\log \pi_1 = \log C + a_2 \log \pi_2 + a_3 \log \pi_3 + \cdots + a_m \log \pi_m
\]

Finally, the equation can be rewritten in matrices form as follows:

\[
\begin{bmatrix}
\log \pi_2 \\
\log \pi_3 \\
\log \pi_4 \\
\end{bmatrix} =
\begin{bmatrix}
\log C \\
2.669 \log h_l/h_u \\
1.2482 \log S_0 \\
\end{bmatrix}
\]

\[
L/h_u = 21335(F_r)^{2.669}(h_l/h_u)(S_0)^{1.2482}
\]
to develop empirical equations like (21) from the numerical data using which the jump location may be predicted without high speed computers.

8. Conclusion
A numerical model is presented to simulate the hydraulic jump numerically in a rectangular channel. Data obtained by laboratory tests for Froude numbers ranging from 2.17 to 7.00 for three different slopes 0.0475, 0.02174, 0 are used to verify the numerical models. The one-dimensional Saint Venant equations are solved numerically to simulate the hydraulic jump. By starting with the computed initial conditions, these equations are solved in time subject to appropriate boundary conditions until a final steady state is reached. The MacCormack’s scheme (second-order accurate both in time and space) was used. As with most of the higher-order methods, high-frequency oscillations were produced near the hydraulic jump. These were smoothed by introducing artificial viscosity. The results obtained from the numerical model were compared with the experimental data. For both the non-zero slopes the jump location was downstream of the location obtained experimentally. The jump location is simulated better at lower Froude numbers and matches closely for Froude numbers 3.83 and less. Based on the jump locations predicted numerically a generalised empirical equation was developed which relates the jump location with the other parameters. The equation is valid for non-zero slopes. Though the equation may not give satisfactory result for other experimental work as the equation has been developed based on a small number of experiments but this is an area where further work is can be done to establish an empirical relationship between the hydraulic jump parameters.

9. References
[1] Safrane K 1929 (English trans. by Barnes DP), Bureau of Reclamation Files, Nos. 37 and 38, Bureau of Reclamation (Denver, Colo)
[2] Bakhmeteff BA and Matzke AE 1936 Trans. 101 630-80.
[3] Bradley JN and Peterka AJ 1957 J. Hydraul. Div. 83(5), 1401-06
[4] Das R, Pal D, Das S and Mazumdar A 2014 Arab. J. Sci. Eng. 39(10) 6995-7002
[5] Silvester R 1964 J. Hydraul. Div. 90(1) 23-55
[6] ASCE Task Force 1964 J. Hydraul. Div. 90(1) 121-47
[7] Rajaratnam, N 1967 Advances in Hydrosct, Academic Press 4 198-280
[8] Garg SP and Sharma HR 1971 J. Hydraul. Div. 97(3) 409-20
[9] Leutheusser HJ and Kartha V 1972 J. Hydraul. Eng. Div. 98(8) 1367-85
[10] Sarma KVN and Newnham DA 1973 Water Resour. (London, England) 139-42
[11] Hager WH and Wanoschek R 1987 J. Hydraul. Res. 25(5) 549-64
[12] Chow VT 1959 Open Channel Hydraulics, McGraw Hills, XVIII (New York, USA)
[13] McCorquodale JA and Khalifa A 1983 J. Hydr. Eng. 109(5) 684-701
[14] Abbott MB, Marshall G and Rodenhius GS 1969 13th Congress IAHR, 1 313-29
[15] Katopodes ND 1984 J. Hydraul. Eng. 110(4) 450-66
[16] Chippada S Ramaswamy B and Wheeler MF1994 Int. J. Num. Methods Eng. 37 1381-97
[17] Hamidreza B, Abolfazl S and Hamidreza V 2015 Arab. J Sci. Eng. 40(2) 381-395
[18] MacCormack RW 1969 Paper 69-354, American Inst. of Aero. and Astro.(Cincinnati, Ohio)
[19] Warming RF and Hyet BJ 1974 J. Comput. Phy. 14(2) 159-79
[20] Anderson DA, Tannehl JC and Fletcher RH 1984 Computational fluid mechanics and heat transfer, Hemisphere (New York)
[21] Jameson A Schmidt W and Turkel E 1981 AIAA 14th Fluid & Plasma Dynamics Conf. AIAA American Inst. of Aero. and Astro.(Palo Alto, Calif.) 81 1259
[22] MacCowan AD 1985 XXIIAHR Cong., Int. Asso. Hydraul. Res. (Melbourne, Australia) 2 50-7
[23] Basco DR 1983/USGS, Water Resour. Invest. Report U.S. Geo. Serv.(Reston) 83-4284
[24] Gharangik AM. And Chaudhry MH 1991 J. Hydraul. Eng.117, 9 1195-1210