FLOOD ANALYSIS IN COMPOUND CHANNEL – COMPARISON OF COMPUTATIONAL AND EXPERIMENTAL RESULTS

T. Vijaya Kumar, Darmendra Tewari
Associate Professor , M.Tech Scholar
Dept. of Civil Engineering,
Delhi Technological University, Pin Code 110042, Delhi, India.

ABSTRACT

Flooding of Indian rivers is a very serious problem, since this greatly affects the normal life of people and the social & financial condition of that place. This problem has been identified by many Researchers in the past like Tominaga and Nezu, Ackers and V.T Chow etc. A good amount of research work has been devoted to find out desirable solution of flooding problem. While flooding, river flows with very high discharge and may overflows over its banks and the excess water reaches to its flood plains. This excess water leads to formation of compound channel. The nature of flow during flooding is generally turbulent in nature. It has been found that the velocity of flow in flood plain is less than that of the velocity of flow in main regular channel. It is a well-known fact that this change in velocity between main channel and flood plain leads to formation of a shear layer. These shear layers produce resistance to flow, which leads to uncertainty in forecasting of flow and increases resistance on channel. Geometric similar model can be utilized to find out necessary information. Generally, one dimensional empirical model can be used to forecasting of flow taking the assumption that flow is uniform in compound channel. However, Compound
Open channel has partially uniform flow due to momentum transfer in sub sections and unexpected change in depth of flow. Hence the analysis of turbulent flow is necessary in this situation. In Past, researchers have used various models to analyze turbulent flow in compound open channels, basically for low development length. Hence, in this study an effort is made to analyze the turbulent flow by Large Eddy Simulation method (LES) to forecast the flow and its resistance on channel. The LES is carried out by taking sufficient development length so that uniform turbulent flow can be developed. The development length is incorporated in the computational domain. It is fact that experimental flow analysis of compound open channels with various hydraulic conditions is very expensive and difficult. Hence, this analysis of flow is done, by using software approaches such as used in Ansys Software.

Keywords: Large eddy simulation, Turbulent structure ,Composite friction factor, compound channel, ANSYS.

1. INTRODUCTION

Rivers are always a symbol of beauty in world. Many People have been living close to the banks of waterways from hundreds of years for the need of sustenance, water, and agribusiness. Despite the fact that, flooding issue in waterways has been a significant issue for person as this harms a much loss of property and lives of people groups and creatures. Henceforth, the rate of event of surges has expanded as of late because of environmental change, over the top human mediation, developing populace on the banks of waterways and industrialization. In this manner, it is required to take certain measures to comprehend flooding circumstances by breaking down the idea driving it. Be that as it may, when stream step by step expands, the water transcends bank and floods to the surge fields. For whatever length of time that the stream profundity of the surge plain is little and not practically identical to profundity of fundamental channel, the mean speed of principle channel is bigger than the surge plain and conveys more release than surge fields. It is basic to examine the stream structures that exist in compound open channels to comprehend the circulation of stream and its factors. The cooperation between the essential longitudinal speed and the auxiliary stream speeds are in charge of non-uniform dissemination of stream factors in a compound open channel stream. This non-uniform circulation of stream factors changes resistance of stream over the wetted border of compound open channel stream.In such circumstances the adjustment in resistance of stream is composite and makes distinction in singular fundamental channel and surge plain resistance been made in the content. From mid eighteenth century numerous exact models are demonstrated to experience the disparities in estimating composite grinding variable and release in compound open channel. Floods happen when principle channel has serious release and this extreme release takes after towards the flood plain fields. The channels framed so are known as compound channels. Numerous down to earth issues in flow designing require precise estimating of flow in

compound open channels. For instance, the water driven reaction to flood counteractive action measures, for example, digging in the fundamental channel and bringing down or smoothing in the floodplains, relies on
upon flow speeds in these regions. It is realized that, neighborhood flow conditions decide the disintegration and testimony rates of residue in the principle channel and floodplains. Therefore, exact expectation of release limit of compound channels is greatly fundamental to suggest in flood relief plans. Waterways are equipped for passing on direct flow until the point when the flow is kept to its primary course. Be that as it may, when flow slowly expands, the water transcends bank and floods to
the flood fields. For whatever length of time that the flow profundity of the flood plain is little and not tantamount to profundity of principle channel, the mean speed of primary channel is bigger than the flood plain and conveys more release than floodplains. The distinction of these flow speeds in both these subsections makes vertical Vortices (as appeared in Figure 1) along the vertical interface of primary channel and flood plain. These Vortices are made because of force and mass trade between flood plain and principle channel, which produces shear constrain and
additional resistance expending additional vitality. Because of the utilization of this additional vitality the expectation of stage release bend ends up noticeably hard to get.

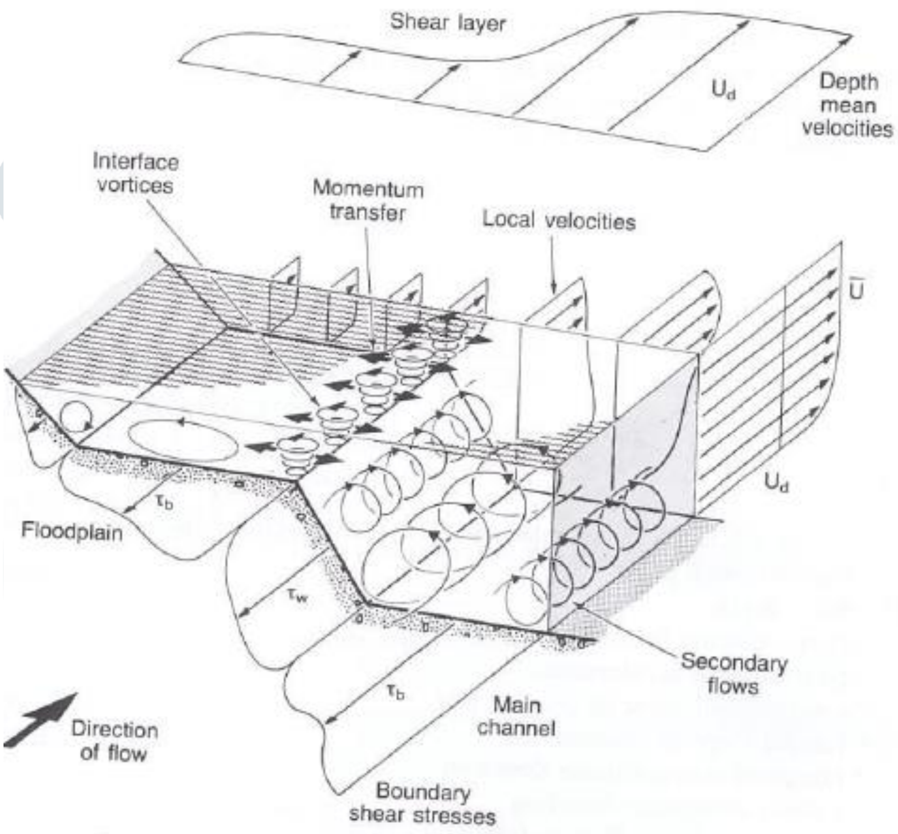

Figure: 1 Details of Hydraulic parameters connected with overbank flow (Knight & Shiono1991)

2. MATERIALS AND METHODS

2.1 Experimental Set up

For present research, one straight experimental compound channel available at Fluid Mechanics and Hydraulics Engineering Laboratory of the Civil Engineering Department at the DTU, Delhi, India is used. The cross-sectional and geometrical parameters are shown in Table 1. The view of experimental compound channels with measuring equipment's from the upstream side is shown in Figure 3. The plan form of the channel, which is having the straight compound channel with equal flood plain at both sides of the main channel as shown in Figure 1. The compound channel is laid inside tilting flume. The flume is equipped with hydraulic jack arrangement. Inside each flume, separate meandering/straight channels are cast using 50 mm thick Perspex sheets. To facilitate fabrication, the whole channel length has been made in blocks of 1.20 m length each. The models thus fabricated have details as: The straight compound channel section has the main channel dimension of 120 mm×140 mm and flood plain width, B = 460 mm. The channel is cast inside a tilting flume of 12 m
long, 600 mm wide, and 600 mm deep. The bed slope of the channel is kept at 0.002.

The Pitot tube was physically rotated with respect to the main stream direction till it recorded the maximum deflection of the manometer reading. A flow direction finder having a least count of 0.1 was used to get the direction of maximum velocity with respect to the longitudinal flow direction. The angle of limb of Pitot tube with longitudinal direction of the channel was noted by the circular scale and pointer arrangement attached to the flow direction meter. The details of experimental parameters for Type-I Compound Channel are shown in Table.

Table 1 Details of experimental parameters for Compound Channel

Sl No	**Item description**	**Specification**
1	Main Channel section	Rectangular
2	Width of the channel (b)	120 mm
3	Full depth of the main channel	140 mm
4	Top width of the Combined (compound) channel	460 mm
5	Slope of the channel	Variable
6	Flume size	0.6 m X 0.6 m X 12 m long

Figure 2: Experimental compound channel

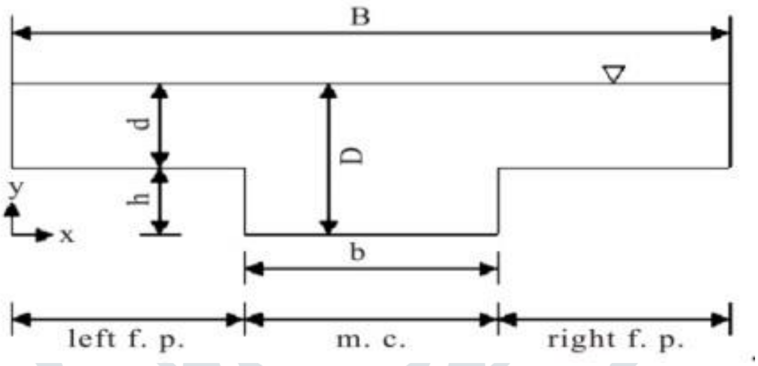

Figure 3: Geometry Setup of compound channel

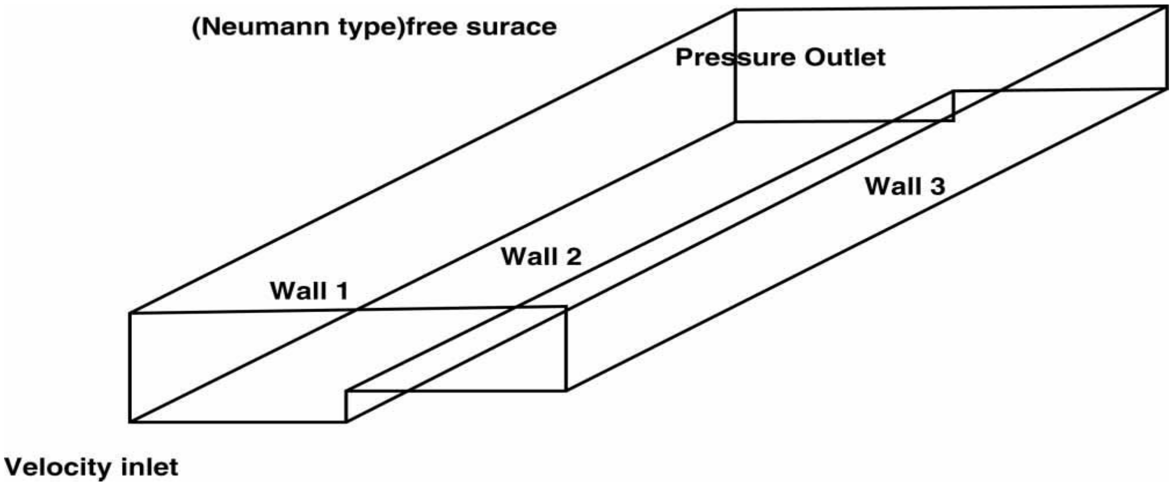

Figure 4: Schematic presentation of geometric alignment and boundary conditions of the Channel.

2.2 Source of Data

The data are collected from research work done in Flood Channel Facility, which is a large scale compound channel facility, available at the laboratory of University of Birmingham, Wallingford. FCF data series A for straight rough and smooth channels, work done by Knight and Demetriou (1983), Atabay (2004) for symmetrical and asymmetrical data series, and Tang (2001) for rough bed and mobile channel data series are used along with experiment work done in Fluid Mechanics Laboratory, DTU Delhi.

2.3 Geometry Set up and Discretization of domain

The fluid flow governing equations (momentum equation, continuity equation) are solved based on the discretization of domain using the Cartesian co-ordinate system. This procedure involves dividing the continuum into finite number of nodes. The CFD computations need a spatial discretization scheme and time marching scheme. Mainly the domain discretization is based on *Finite element, Finite Volume and Finite Difference Method*. Finite Element method is based on dividing the domain into elements. The numerical solution can be obtained in this method by integrating the shape function and weighted factor in an appropriate domain. This method is suitable with respect to both structured and unstructured mesh. The application of Finite Volume method needs dividing the domain into finite number of volumes. Here the specified variables are calculated by solving the discreitized equation in
the center of the cell. This method is developed by taking conservation law in to account. Finite Volume method is suitable for applying in unstructured domain. Finite Difference method is based on Taylor's series approximation. This method is more suitable for regular domain.

3 RESULTS & DISCUSSION

3.1 Comparison of experimental and Simulation Results

The numerical simulation is carried out by using ANSYS-CFX solver and the numerical results are compared with the experimental results. The results are tabulated in Table 3.1. Here mean bulk velocity is calculated using the formulation:

$$W_b = \frac{\int wdA}{A}$$

Where, W_b = Bulk Velocity along Stream-line of flow. w = streamline velocity at any point, A = Cross section area of the channel. The composite Manning's friction factor is calculated from Manning's equation.

Table 2 Comparison of the experimental and simulation results

Type	Max Velocity W max (m/s)	Discharge	Mean Bulk Velocity W_b (m/s)	Composite Mannings n	Shear Velocity u* (m/s)
S-1 (Tomminaga and Nezu 1991)	0.409	0.00738	0.368	0.011383	0.0161
LES simulation	0.4074	0.00732	0.366	0.0113819	0.0162

The table shows that, the results obtained from LES simulation are in good agreement with case S-1 of Tominaga and Nezu (1991).

3.2 Velocity Distribution

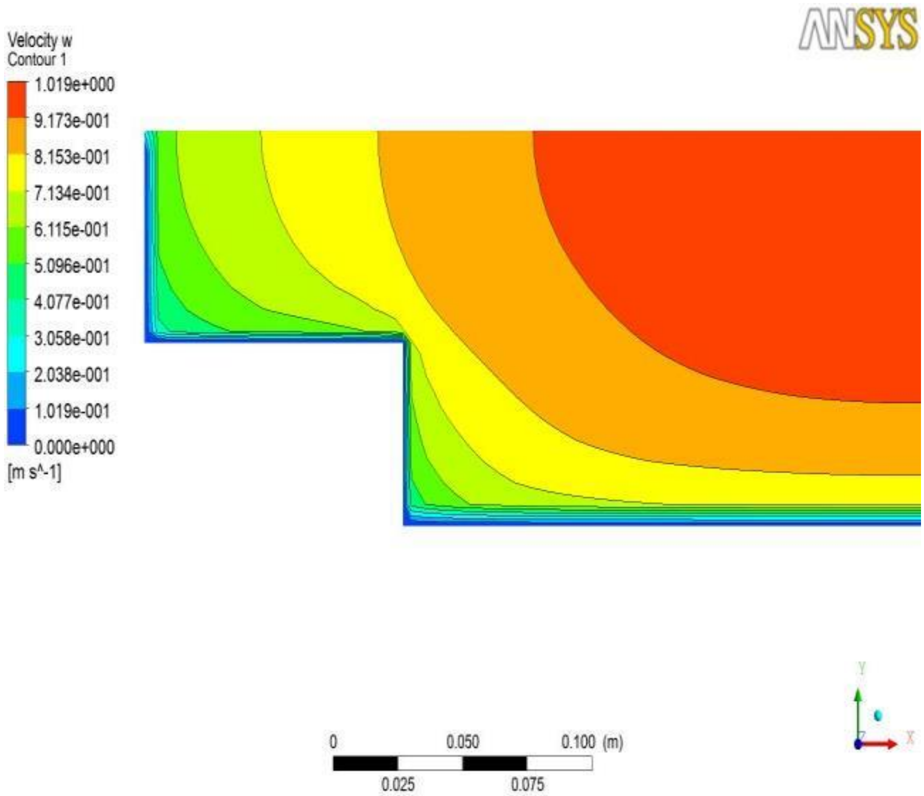

Figure 5: Mean Velocity distribution of LES Simulation

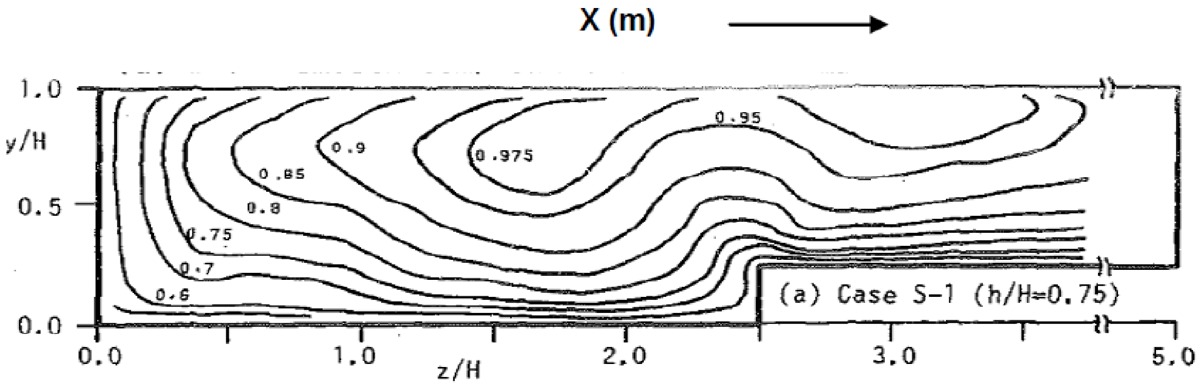

Figure 6 : Mean Velocity distribution of experiment. (Tominaga and Nezu 1991)

The isovel -lines of the non-dimensional stream-wise velocity W (z) are computed by LES method as shown in Figure 5. It shows from simulation that maximum velocity is 0.4049 m/s and can be observed near centerline of channel at approximately 0.057m from centerline of the channel.

The bulk velocity is 0.367 m/s. Isovel lines bulge significantly upward in the vicinity of the junction edge along the flow. The patterns of the isovel lines are convincingly followed by LES simulation results with the experimental results of Tominaga and Nezu (1991) as shown in Figure 6. The reason of this bulge is the decelerated region on the both side of the junction region of main channel. The decelerated region is created because of low-momentum transport due to secondary current away from the wall. This causes the bulge in the main channel and flood plain interface due to high momentum transport by secondary current. Consequently, primary velocity is directly affected by momentum transport due to secondary current. Figure 6 shows the distribution of non- dimensional mean velocity.

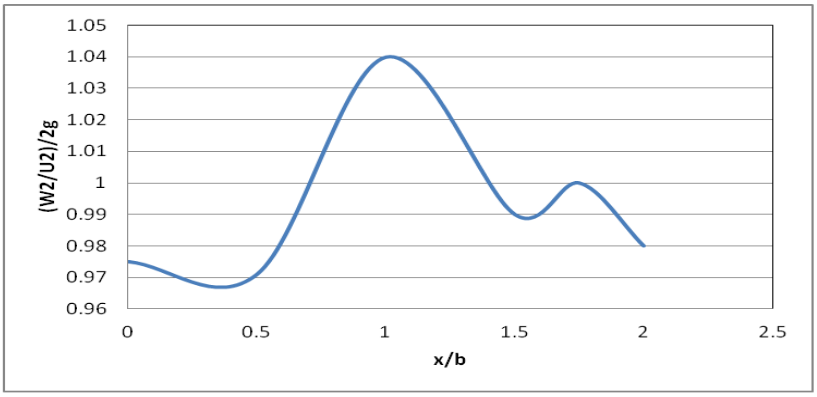

Figure 6: The distribution of depth averaged velocity head.

3.3 Bed Shear Stress

The distribution of the non-dimensional bed shear stress (avg.τ /τ) obtained after simulation is presented in Figure 7. The pattern of distribution is found to be distributed evenly with the experimental results. The actual distribution of bed shear stress in experimental results attains two peaks one at flood plain and other at the main channel. The simulation result has also attained the same pattern and which show high degree of accuracy of simulation. The distribution shows that the peaks can be observed both side of the junction of the main channel and flood plain. The average bed shear distribution in main channel is found to be lesser than the flood plain.

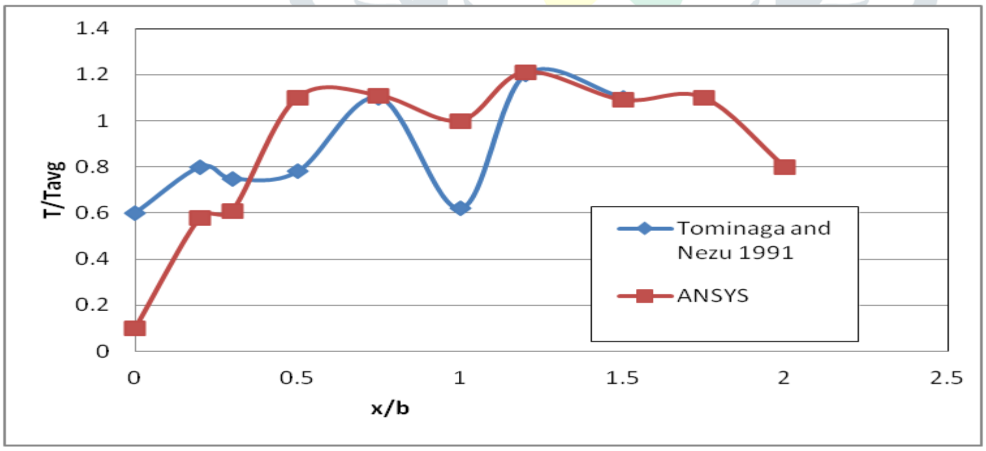

Figure 7 : The distribution of non-dimensional bed shear stress

3.5 Velocity contours

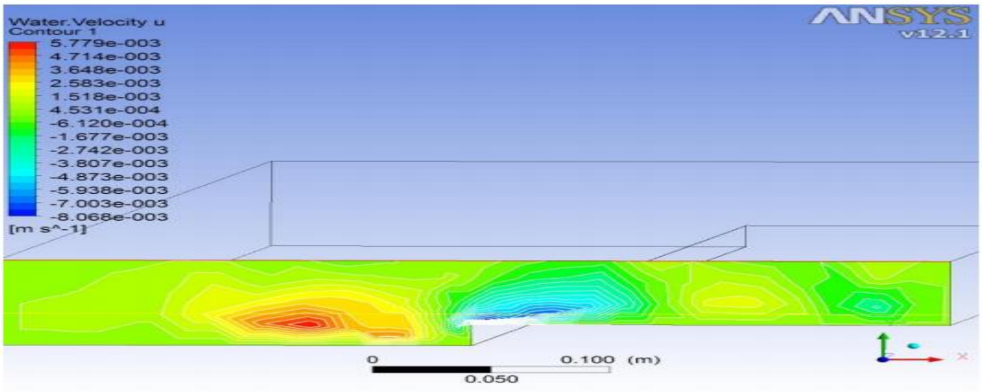

Figure 8 : Stream-wise non-dimensional averaged secondary velocity contours

It can be observed a pair of secondary currents on the both side of the junction of the main channel and flood plain. These currents can be regarded as longitudinal vortex as mentioned by Tominaga and Nezu (1991). The vortex in flood plain reaches the free surface. The mean secondary velocity contours show circulation at the main channel corner, main channel flood plain interface, at the corner of flood plain as shown in Figure 8. Which is quiet convincing with experimental secondary current vectors as shown in the Figure 8. These resemblances of result have significant contribution on the distribution of average velocity. The large counter rotating secondary structure produces usual velocity dip and has maximum impact on stream wise velocity. This counters rotating flow structure creates resistance and reduce the average velocity, thus discharge. Because of this structure, it is difficult construct a one-dimensional model**.**

4. CONCLUSION

Based on analysis and discussions of this study following conclusions can be drawn. The conclusions from the present work are as follows:

LES simulation results are presented to show the velocity distribution and secondary current, momentum transfer from main channel to flood plain and vis-versa in an asymmetric compound channel. The discharge and composite friction factor found from the LES simulations are also in good agreement with experimental results.

Different discharge and composite friction factor prediction methods are studied. These methods are applied to the published data of compound channels with different hydraulic conditions.

The methods are found to give good results to some compound channels whereas fail to give good results for compound channels of other geometry and hydraulic conditions.

Simulation by LES is done for a compound channel of single flow depth. Simulating LES for different hydraulic condition for different compound channels are arduous and computationally expensive.

5. REFERENCES

1. A. Tominaga and J. Nezu, “Turbulent Structure in Com- pound Open Channel Flow,” ASCE Journal of Hydraulic Engineering, Vol. 117, No. 1, 1991, pp. 21-41.
2. Cokljat D. (1993).Turbulence models for non-circular ducts and channels. PhD Thesis. City University, London.
3.Colebatch GT.(1941). Model tests on the Lawrence Canal roughness coefficients. J. Inst. Civil Eng.(Australia), 13(2),27–32.
4. Chow VT. (1959). Open-channel hydraulics. New York: Mc. Graw-Hill Book Co.
5. Krishna pan BG. and Lau YL. (1986). Turbulence modeling of floodplain flows. Journal of Hydraulic Engineering, ASCE. 112(4), pp. 251-266.
6. Krishnamurthy M, and Christensen BA. (1972). Equivalent roughness for Shallow channels. J. Hydraulics Div., 98(12), pp. 2257–2263.
7. Knight DW, and Hamed ME. (1984), Boundary Shear in Symmetrical Compound Channels. J. Hydraul. Eng., ASCE, 110, pp.1412-143

8. Knight D. W. (1999). Flow mechanics and sediment transport in compound channels [J]. International Journal of Sediment Research, 14(2): 217-236.
9. Myers WRC, (1987). Velocity and Discharge in Compound Channels, Jr. Hydr. Engg, ASCE, Vol.113, No.6, pp.753-766.
10. Rajaratnam N, and Ahmadi RM. (1979). Interaction between Main Channel and Flood Plain Flows. J. of Hydr. Div., ASCE, 105(5), pp. 573-588.
11. Sellin RJH. (1964). A laboratory investigation into the interaction between flow in the channel of a river and that of its floodplains. Le Houille Blanche 7, pp. 793-801
12. Shino, K. & Knight, D. W. (1991) Turbulent Open-Channel Flows With Variable Depth Across the Channel. Journal of Fluid Mechanics
13.Shiono K, Al-Romaih JS, and Knight DW. (1999) Stage- Discharge Assessment in Compound Meandering Channels. J. Hydraul. Eng., ASCE, 125(1), pp.66-77.

14 Tominaga A and Nezu I. (1991). Turbulent structure in compound open channel flows. J. Hydraul. Eng., ASCE, 117(1).
15. Van Prooijen BC, Battjes JA, and Uijttewaal WSJ. (2005). Momentum exchange in straight uniform compound channel flow. Hydraul. Eng., ASCE, 131(3), pp.175 83.
16. Walid HS, and Shyam SS. (1998). An artificial neural network for non-iterative calculation of the friction factor in pipeline flow. Compute Electron Agriculture, 21, pp.219–28.