3D modelling of Porcupine River Training Works

Riddick Kakati

PhD. Scholar, IIT Guwahati

ABSTRACT

Porcupines are popularly used river training works that are deployed to retard the flow velocity of rivers thereby preventing river bank erosions. They have already been used in major Indian Rivers such as The Brahmaputra and The Ganga. However, it has been seen that these structures often fail to perform adequately during periods of high floods and are often washed away. This simulation aims to map the zone of influence of scaled down porcupine structures placed in a laboratory flume of IIT Guwahati assuming steady, incompressible and turbulent flow field within OpenFOAM. Results obtained were then compared with experimental results.

1. Introduction

River training works refers to various measures adopted on a river to direct and guide the river flow, to train and regulate the river bed or to increase the low water depth (Punmia B.C., 2009). They are primarily categorized as permeable and impermeable river training works. Permeable river training works allow partial entrainment of flow through them, thereby reducing flow velocity e.g., Porcupines (Figure 1 a). Impermeable river training works on the other hand obstruct the flow completely e.g., Impermeable Groynes (Figure 1 b).

Figure 1: Permeable River training works (Left) and Impermeable River training works (Right)

Due to partial obstruction, permeable river training works prevent formation of scour holes, which otherwise would result in structural instability. However, during periods of high flow, these permeable structures are often washed away. To design an optimum layout for river training, it thus becomes necessary to understand the flow field generated after placement of river training works. Mathematical modelling studies can be conducted to generate these flow fields. However, due to several assumptions, the results obtained from these studies need to be validated with experimental results.

2. Problem Statement

Due to several advantages of porcupine structures such as low cost, less installation time and absence of scour hole, porcupines are very widely used in Indian rivers. However, at high floods, these structures are often washed away, leading to severe bank erosions at the locations which were intended to be protected by them. Study of the flow fields generated by these structures before deploying them will help in proper designing of these structures. The objective here is to map the zone of influence generated by a single row of porcupine structures. This will help in predicting whether a particular layout of these structures placed in a river will be able to protect the desired bank from erosion. Also new modified layouts can be tested before actual implementation. Thus, an optimum layout can be designed as per site requirements.

3. Governing Equations

The basic governing equations solved are the continuity equation (Equation 1) and the three-dimensional Navier Stokes Equations (Equation 2). In this case the following assumptions were applied:

- Flow is incompressible
- Flow is steady
- Properties of fluid (clear water in this case) is uniform throughout
- Temperature effects on viscosity of water are ignored
- No slip at bottom boundary of flow domain and solid boundaries of the porcupine structures
- Bed and banks of the flow domain are non- erodible
- Turbulence was modelled using kOmega SST Turbulence model (Equation 3)

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \rightarrow Equation\ 1$$

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\rightarrow Equation\ 2\ (a)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\rightarrow Equation\ 2\ (b)$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

$$\rightarrow Equation\ 2\ (c)$$

Turbulence specific dissipation rate equation:

$$\frac{D}{Dt}(\rho\omega)=\nabla\cdot(\rho D_\omega \nabla\omega)+\frac{\rho\gamma G}{\nu}-\frac{2}{3}\rho\gamma\omega(\nabla\cdot\omega)-\rho\beta\omega^2-\rho(F_1-1)CD_{k\omega}+S_\omega$$

$$\rightarrow Equation\ 3\ (a)$$

Turbulence kinetic energy equation:

$$\frac{D}{Dt}(\rho k)=\nabla\cdot(\rho D_k \nabla k)+\rho G-\frac{2}{3}\rho k(\nabla\cdot u)-\rho\beta^*\omega k+S_k$$

$$\rightarrow Equation\ 3\ (b)$$

Turbulence viscosity equation:

$$\nu_t=a_1\frac{k}{\max(a_1\omega, b_1F_{23}S)}$$

$$\rightarrow Equation\ 3\ (c)$$

These equations are solved after numerical discretization using Finite Volume Approach and SIMPLE algorithm.

4. Simulation Procedure

A 20 m long and 1 m wide straight rectangular channel setup of Fluvial Hydro-Ecological Laboratory of IIT Guwahati was replicated in OpenFOAM. Scaled down models of porcupines (1:50) were placed at the center of the flume. Velocity fields were then generated to study the impact of these structures. Validation with experimental results was done by comparing the velocity profiles at upstream, downstream and an unobstructed location.

4.1 Geometry and Mesh

In this case, the geometry can be categorized as flow geometry and object geometry. The flow geometry consisted of a hollow parallelopiped with length 20 m and width 1 m. The mesh was generated using 100 divisions in flow direction (x-axis) and 10 divisions in y and z axes respectively. This was created by using the blockMesh utility of OpenFOAM. Scaled down porcupine structures (1:50) were then created and then generated as “stl” objects in FreeCAD. Then these objects were incorporated into the geometry by using snappyHexMesh utility of OpenFOAM. Finer mesh was generated at the locations where the porcupine structures were placed. The files to set parameters for blockMesh and snappyHexMesh can be found inside the system folder of the case directory. The resulting geometry layout is shown in Figure 2 which can be viewed after execution of snappyHexMesh in ParaView.

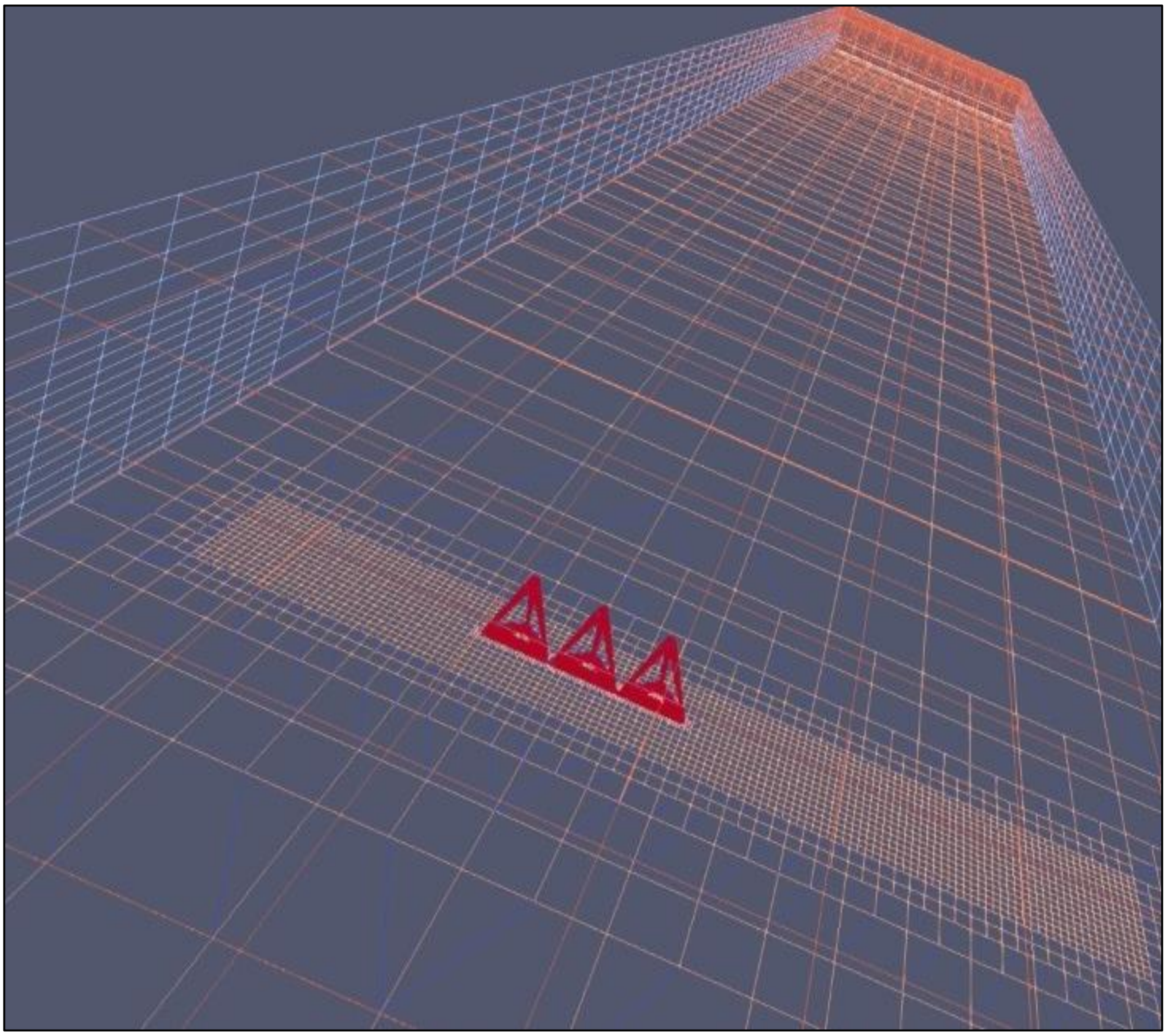


Figure 2: Geometry preview in ParaView

4.2 Initial and Boundary Conditions

The various initial conditions used are shown in Table 1 below:

Table 1: Initial Conditions used for Simulation

Sl. No.	Face	Quantity	Value (S.I. Units)
1	-	Velocity	1.15×10^{-1}
2	-	Pressure	0
3	-	Turbulent Kinetic Energy	1.98375×10^{-4}
4	-	Turbulent Omega	5.71×10^{-1}
5	-	Dynamic Viscosity	1.5×10^{-5}

In addition to these, the bottom wall of flow domain as well as the members of the porcupine structure had been assigned no slip boundary condition and the logarithmic velocity profile due to boundary layer formation was replicated using wall functions.

4.3 Solver

The simpleFoam solver in OpenFOAM was most suitable for this case due to the assumptions considered viz., steady and incompressible flow. The simpleFoam solver is an iterative solver that utilizes "Semi- Implicit Method for Pressure- Linked Equations (SIMPLE)" algorithm to solve the Navier Stokes Equation. Turbulence was incorporated into the model using kOmega SST model.

Module Name	simpleFoam
Simulation Time	36000 seconds
Fluid type	Newtonian
Viscosity	1.5e-05
Turbulence Model	kOmegaSST
Simulation Type	RAS
Computer Specifications	HP Z200 SFF Workstation Intel Xeon Processor X3430 2.40 GHz, 8MB cache, 1333 MHz memory, 8 GB RAM
Operating system and softwares	Linux Ubuntu 20.04 with openFOAM v8

5. Results and Discussions

After the simulations have been performed, the results have been extracted using two methods. Velocity contours were generated using ParaView to extract slices and then using python script along with matplotlib to plot them. Velocity profiles were generated using the probe function in OpenFOAM to extract the field values at the probe locations, which were then plotted using python script along with matplotlib. The results obtained in OpenFOAM simulation were compared with experimental results conducted at the Fluvial Hydro- Ecological Laboratory of IIT Guwahati (Figure 3). The experiments were conducted with reference to the experiments conducted by Aamir and Sharma (2015). Three- dimensional point velocity measurements were taken in the laboratory using Nortek Vectrino Acoustic Doppler Velocimeter (ADV).

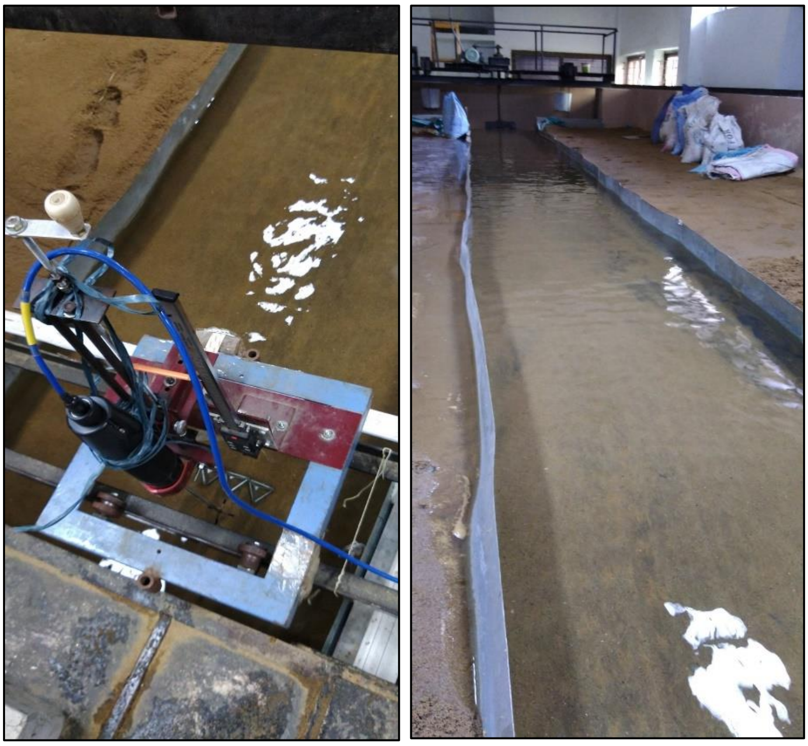


Figure 3: Velocity measurements being taken in the laboratory using Acoustic Doppler Velocimeter (Left) and the experimental flume setup of dimensions 20 m x 1 m (Right)

Firstly, the results obtained from model were validated with experimental results. Corresponding probe locations in model and ADV measurements in experiment are shown in Figure 4.

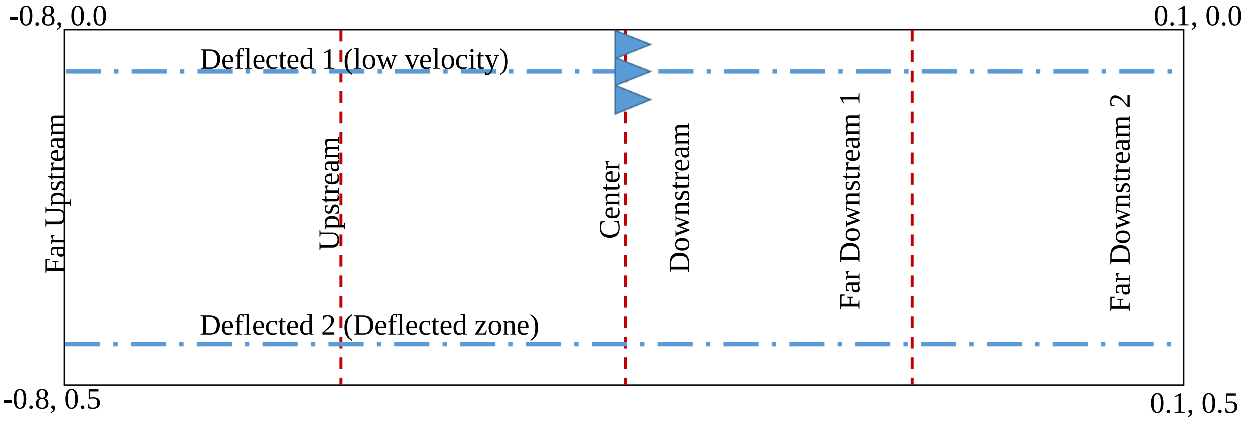


Figure 4: Test section along with notations and coordinates

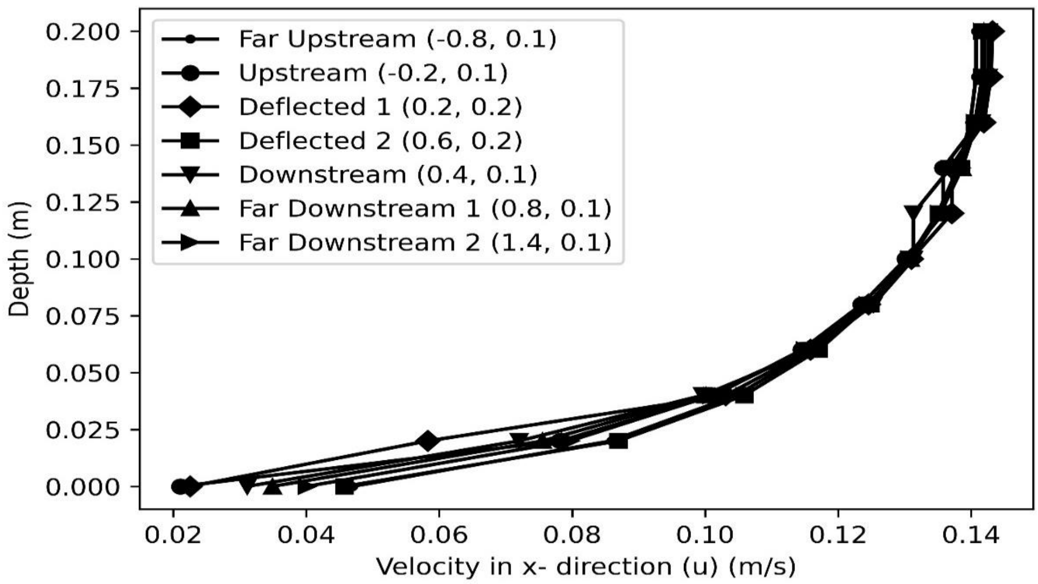
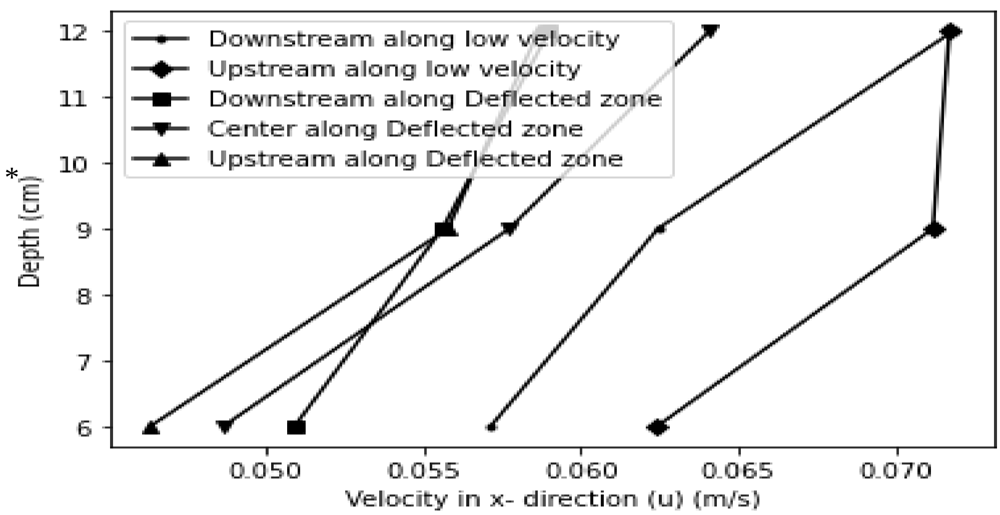


Figure 5: Numerical Model Results



* Depths with respect to ADV location. Actual measurement location (sampling volume) lies 5 cm below this location

Figure 6: Experimental Results

Comparing the modelling results (Figure 5) with experimental results (Figure 6), it can be observed that the impact of placing single row of porcupine structure in the flow field is very less. Along deflected zone in experimental results (Figure 6) are discarded because this was very close to the wall of the flume, hence wall effects came into picture. Despite that it can be seen that in the absence of porcupine structures, there is no change in velocity at upstream and downstream. The variation of modelling results from experimental results in terms of velocity profile shape and minor deviations are attributed to:

1. The assumptions in the numerical model
2. The time averaging of experimental velocities from instantaneous velocity measurements

Finally, the zone of influence was plotted and is shown in Figure 7. Here also it is observed that single row of porcupine structures is not efficient in the case considered.

(a).
(b).

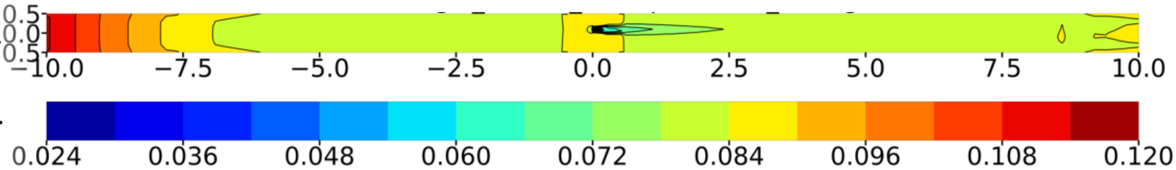


Figure 7: (a). Velocity contour map along with coordinates (Scaled to flume dimensions)

(b). Velocity (m/s) corresponding to colour for contour map in (a).

Conclusion

In this study, the influence of porcupine on a flow field was examined using three- dimensional numerical model OpenFOAM. The results obtained from numerical modelling were validated using experimental results. It was observed that single row of porcupine placed in a flow field is not very efficient in reducing and diverting flow velocity. In this case, a proper understanding of the flow field generated after placing porcupine structures enabled us to predict the behaviour of the structure after submergence. Thus, carrying out numerical modelling studies before implementing river training works will be helpful in designing an optimum site-specific layout.

References

Aamir, M., & Sharma, N. (2015). Riverbank protection with Porcupine systems: Development of rational design methodology. *ISH Journal of Hydraulic Engineering*.

Cengel, Y. A., & Cimbala, J. M. (2010). *Fluid Mechanics Fundamentals and Applications (in SI Units).* New Delhi: Tata McGraw Hill Education Private Limited.

Greenshields, C. J. (2020). *OpenFOAM User Guide.* The OpenFOAM Foundation.

Punmia, B., Lal, P. B., Jain, A. K., & Jain, A. K. (2009). *Irrigation and Water Power Engineering.* New Delhi: Laxmi Publications (P) Limited.