Groyne Effect on Flow Modulation Over Curved Channels

Abhirami V. S.
Dept. of Civil Engineering,
Government Engineering College
BartonHill
abhishalu2002@gmail.com

Aswathy K. Sundaram
Dept. of Civil Engineering,
Government Engineering College
BartonHill
aks2000115@gmail.com

Sneha A. P.
Dept. of Civil Engineering,
Government Engineering College
BartonHill
snehaajith001@gmail.com

Vishnu Priya A. S.
Dept. of Civil Engineering,
Government Engineering College
BartonHill
vishnupriyasumi2001@gmail.com

Dr. Wilbert R.
Dept. of Civil Engineering,
Government Engineering College
BartonHill
wilbertcet@gmail.com

ABSTRACT

River meanders are key features on the earth surface. Meander migration reshapes land and rejuvenates flood plain. Curvature effects due to meandering leads to changes in velocity patterns, pressure gradient, superelevation of free surface and formation of secondary flow. Transverse convection of momentum by secondary flow give rise to considerable deformations of the main velocity distribution such that main velocity maximum tends to shift from inner to outer bend. This leads to outer bank erosion and undermining. All these parameters are studied using numerical simulation approach with the aid of Computational Fluid Dynamics (CFD)-ANSYS FLUENT software.

As a contribution to the development of mathematical model of flow and bed topography in river bends, steady flow in strongly curved open channel flow is analysed mathematically by solving the complete Navier-Stokes equation. For this purpose, we create a 2D model of straight and strongly curved open channel flow inbuilt with groynes and do the necessary simulations using ANSYS. Groynes are hydraulic structures constructed across the river to control the flow away from banks to minimize erosion. Favourable variations of hydraulic parameters like pressure and velocity around a groyne for maximum protection of bank must be found out.

Keywords - Computational Fluid Dynamics; ANSYS FLUENT; Groyne; Numerical simulation;2D Model.

1. INTRODUCTION

1.1 General

One of the most eye-catching features of rivers is their meandering pattern. The important features of rivers are the turbulence of the flow and the often-complex geometry of the river bed. This paper reports the results of a detailed investigation of the key features of river bend

flows. In view of the high complexity of this kind of flows, it has been chosen to investigate scaled models of river flows. It is for this reason that the more general term of curved open-channel flows is used. The research has been conducted by means of detailed computer models and by so-called Reynolds-Averaged Navier Stokes (RANS) model.

1.2 Ansys Fluent

Computational Fluid Dynamics (CFD) is a tool used to analyse condition; it uses set of mathematical equation to be solved in computer for obtaining the results. The basic idea behind using CFD is to analyse the fluid flow in detail and this done by converting the continuous flow equation for the region of interest into the discretised mathematical sets of equation, by applying various boundary condition which are know in advance. Growing use of CFD as analysis tool is due to its advantage of precise modelling, accurate results and less computational cost.

One of the CFD solver used for analysing complex flow is ANSYS Fluent. ANSYS Fluent is incorporated within the ANSYS Workbench platform. ANSYS Fluent provide the option for geometry construction and meshing and can also solve the unstructured mesh, which is useful to mesh around complex geometries. It supports various mesh type like quadrilateral, triangular, tetrahedral, hexahedral, and polyhedral. The software also provides the option of dynamic coarsening or refinement of the mesh size, which ultimately helps in obtaining the accurate results or on reducing the run time required for analysis.

ANSYS Fluent can be used for all types of models and flow like transient or steady state flow, incompressible and compressible flow, turbulent and laminar flow. It provides the option to the user to adapt from various type of models that is applicable for the given conditions. In the real-life situation, most of the open channel flows are turbulent in the nature so ANSYS Fluent more turbulent models like one equation model or various two equation model. ANSYS Fluent also provides tools which enables the user to choose the material either it of fluid flowing or the material from which channel is constructed, this allows more realistic conditions. Number of iterations can be varied according to the experience that in how many iterations model will get converged. ANSYS Fluent has its own embedded post-processing tools which enables the user to obtain various contours, velocity vectors, streamlines and various other sets of representative results that are useful for analysis of fluid flow.
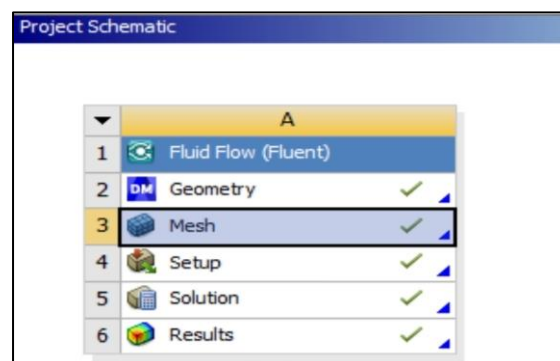

Fig 1: Project Schematic in ANSYS workbench

2. OBJECTIVES

- To study flow and turbulence around curved path.
- To study flow velocity amplification over the curved boundaries.
- To evaluate the effectiveness of groynes on bank protection.
- To find out most appropriate layout of the series of groynes.
- To obtain variations in velocity and other parameters around different layout of groynes.

3. METHODOLOGY

3.1 Physical model

The model was created using ANSYS Fluent software. A channel with 193-degree bend and 5m width was created and different arrangements of groynes were provided by trial and error method.
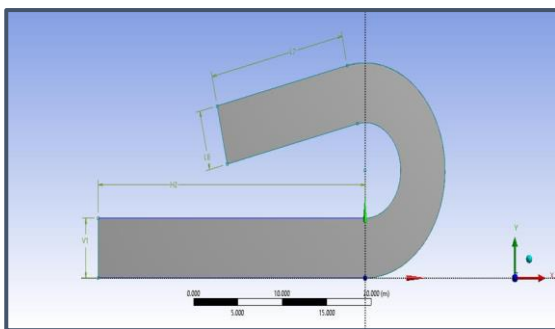
Fig 2: 193 degree strongly curved channel

3.2 Numerical Method

For analysing these models through numerical model, ANSYS Fluent is used. In numerical simulation of fluid flow undergoes following three steps:

(a) Pre-Processing

- Construction of geometry
- Meshing and Domain designation
- Selecting the model to be used for solving, based on flow type
- Providing initial condition and defining boundary condition

(b) Solver

- To converge the solution time steps and number of iterations is selected accordingly

(c) Post processing

- After calculation is completed various results in the form of contour and charts are analysed.

3.3 Standard k-epsilon model

The basic assumption in this model is that the flow in fully turbulent and the molecular viscosity has negligible effect. It is a two-equation model, in which turbulence is quantified with the help of two parameters:

a) Turbulent Kinetic Energy (k)
b) Turbulent Dissipation Rate (Є)

The model transport equation for k is derived from the exact equation and the model transport equation for ϵ is obtained using physical reasoning.

i)Turbulence Kinetic Energy(k)-

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial t}$$

$$= \frac{\partial|(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}|}{\partial x_j} + G_k + G_b - \rho\varepsilon - Y_m + S_k$$

ii)Turbulence Dissipation Energy(ε)-

$$\frac{\partial)\rho\varepsilon}{\partial_t} + \frac{\partial(\rho\varepsilon u_i)}{\partial_t} = \frac{\partial|(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial k}{\partial x_j}|}{\partial x_j} + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k +$$

$$C_3 G_b) - C_{2\epsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

G_k - turbulence kinetic energy due to the mean velocity gradients,

G_b - generation of turbulence kinetic energy due to buoyancy.

Y_m - fluctuating dilatation in compressible turbulence to the overall dissipation rate.

$C_{1\varepsilon}$, $C_{2\epsilon}$ and $C_{3\varepsilon}$ are constants.

σ_k and σ_ε are the turbulent Prandtl numbers for k and ε respectively.

3.4 Pre-Processing

3.4.1 Geometry Setup

In ANSYS workbench there is design modeller in which geometry can be constructed.

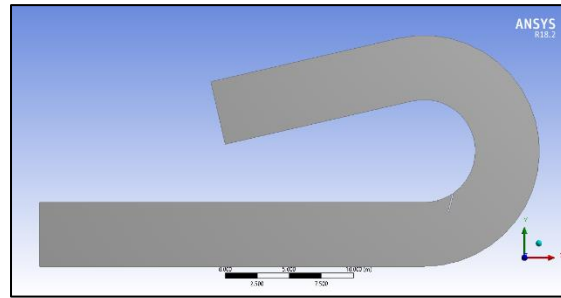| ENTITY	VALUE
Width of channel	5m
Length of Groynes	1.5m
Width of Groynes	0.1m
Inclination of Groynes	30 degree

Fig 3: Geometry of 193-degree bend with one groyne

3.4.2 Meshing

Meshing tool is available in ANSYS Workbench so meshing of various model is done through it only. The desired physical preference and accuracy required will alters the mesh type and its size. In the meshing step, we also name the section which will act as different boundary parts of the model like inlet, outlet, top surface, bed etc

OBJECT NAME	MESH
Physics Preference	CFD
Solver Preference	Fluent
Mesh	Fine
Element order	Linear

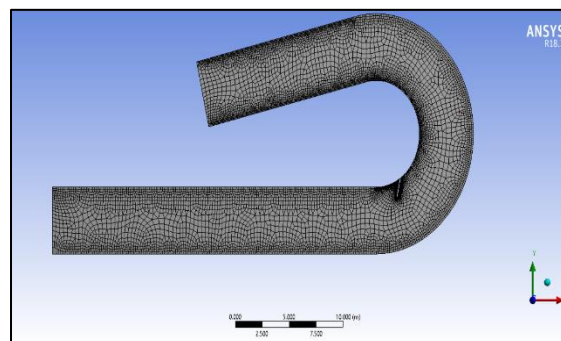

Fig 4: Meshing of the face created with one groyne

3.5 Solver

Solver is the significant step which determines the characteristics of the result. It converges the solution time steps and number of iterations is given in this step. In fluent setup we assign data like input variables boundary conditions type of model etc.

3.5.1 Fluent Setup

It is the part of software where we provide the information, like input variables, boundary conditions, type of model to be used, to the software for calculation purpose. In this, the first step is to provide details like value and direction of the gravity. Solver details like steady or transient time analysis, pressure based or density-based analysis. Units of various parameter can also be changed from this interface.

OBJECT NAME	SETUP
Model	k-epsilon
Material	Water
Boundary Conditions	• Inlet velocity=3m/s • Outlet pressure=0
No of iterations	1000

3.5.2 Solution

Next step is to initialize the solution. Total number of iterations are always more than 500 and it is found that solution always gets converged in less than 500 iterations. Now run the calculation using solution branch in the tree

to perform solution-related activities. This is the step in which solution methods for different computations are defined and also monitors for control. Gradients of solution variables are required in order to evaluate diffusive fluxes, velocity derivatives and for higher order discretization schemes.
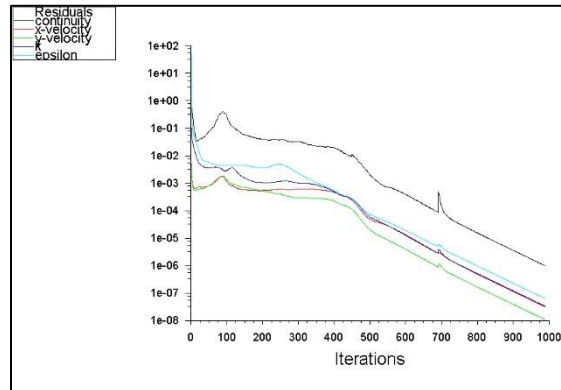

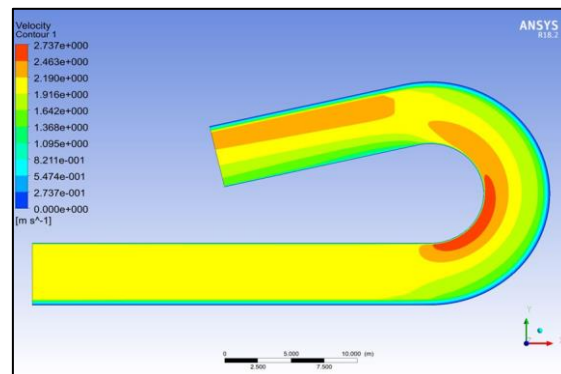Fig 5: Solution Convergence in ANSYS Software

Fig 6: Velocity contour without groyne.

Inclination of attracting groynes is towards downstream; therefore they attract the flow towards themselves causing formation of scour hole near to the bank.

Inclination of the repelling groynes is towards upstream direction due to this they repel the water mass away from the protected bank.

3.6 Post Processing

The results are obtained in the form of contour, graphs charts etc, where the user can study analyse and compare the results. Also, different sections can be constructed and the pressure and velocity variations at those sections can be analysed. The results of simulation will be displayed in Ansys Fluent.

As fluid (water) approaches groin, which is placed at a particular orientation, flow separation occurs and consequently, it leads to the formation of vorticity point. At the point of vorticity, both the hydraulic parameters (velocity and pressure) are varied that is a low pressure and high velocity region is formed.

As velocity is increased, an increase in drag force occurs, which in turn accelerates the turbulence of water, which carries away large amount of sediment along with fluid. This leads to acceleration of erosion phenomenon in riverine systems. Thus the orientation of groin, when placed from river bank, has a significant impact on river flow, wherein its orientation of groin can influence the variation of hydraulic parameters. Out of various inclinations, at optimum inclination, variation of hydraulic parameters (of low velocity and high pressure), such that it satisfies the minimum erosion condition, can be observed.

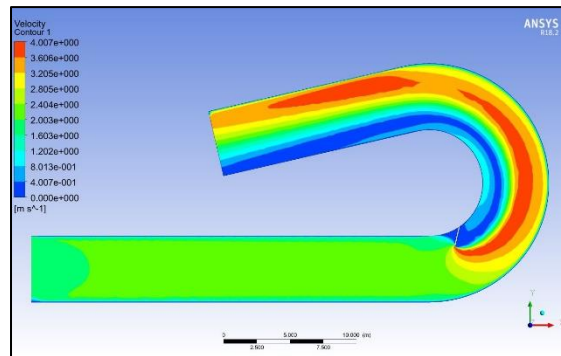The variation of the hydrodynamic performance at varying distance and numbers is studied.

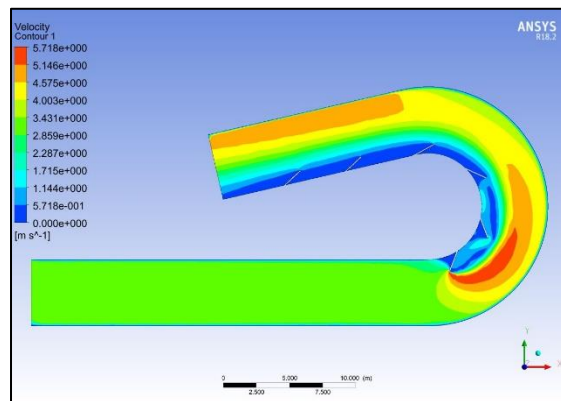Fig 7: Velocity contour with one groyne.

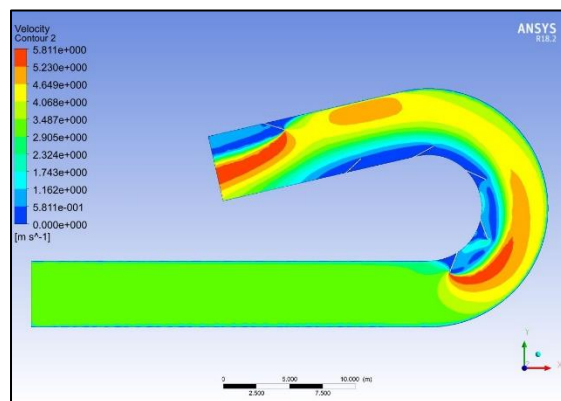Fig 8: Velocity contour with 6 deflecting groynes in the inner bank

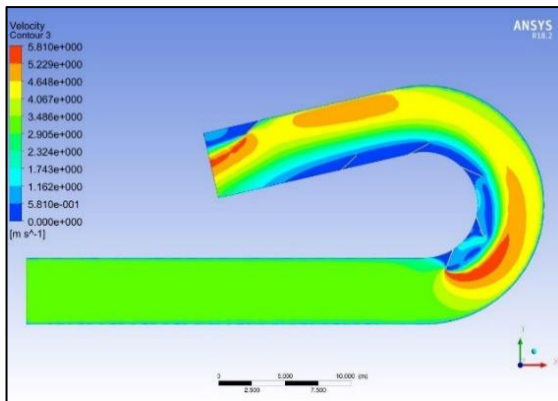Fig 9: Velocity contour with 6 deflecting groynes.

4. RESULTS

Fig 10: Velocity contour with 5 deflecting groynes in the inner bank and 1 attracting groyne in outer bank
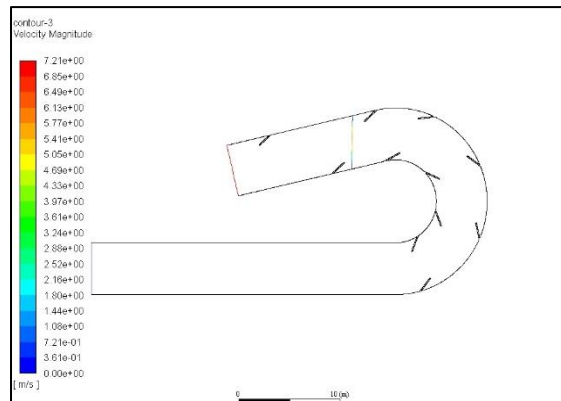

Fig 11: Velocity contour with 5 deflecting groynes in the inner bank and 2 attracting groynes in the outer bank

Fig 12: Velocity contour with 5 deflecting groynes in the inner bank and 6 attracting groynes in the outer bank.

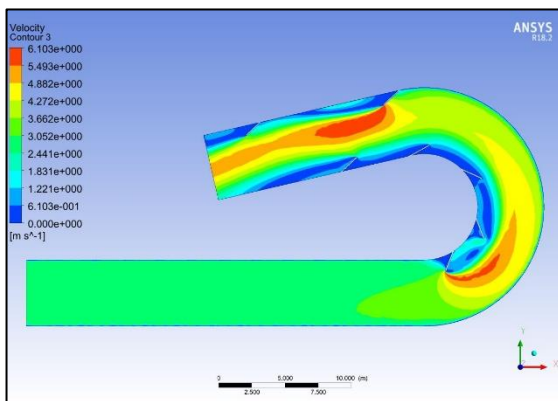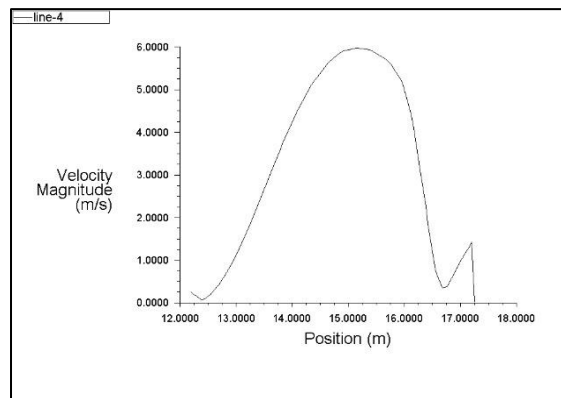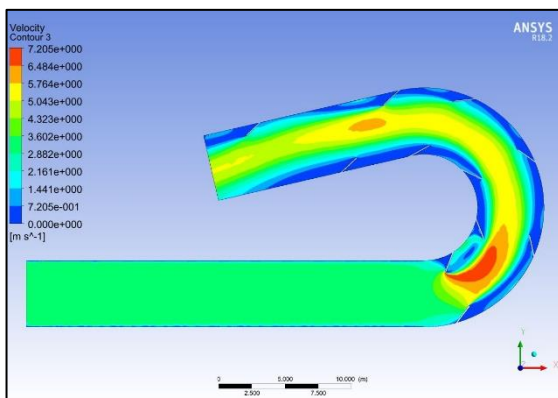Fig 13: Geometry of velocity plot for a line joining (25.75,17.25) and (25.75,12.19)

Fig 14: Velocity plot for a line joining (25.75,17.25) and (25.75,12.19)

CONCLUSION

In this study, the flow pattern around the groin field has been mapped through the study conducted on ANSYS FLUENT software. The study focuses on the representing hydraulic parameters of fluid flow (pressure and velocity) to obtain optimum groin parameters (with varying length of groin, spacing and type) through numerical simulation. The simulation has been initiated with single groin from which the optimum spacing of groin was found out. By post processing the simulation results, the conclusions arrived at are:

- In the curved part of the flume, the centripetal forces give rise to secondary flow cells. At the entry of the bend, the highest velocities are found near the inner bank of the flume. This is explained by the suddenly strong favourable pressure gradient at the inner bank.

- Effect of one groyne has negative impact on both inner and outer bank. A single groyne may not be adequate for protecting the bank against strong and constant water flow, as it may only divert the water flow in a specific direction, creating erosion problems downstream of the structure.

- Series of groynes placed at the inner bank reduced the impact of soil erosion.

- The maximum velocity is observed at the tip of the groin where maximum turbulence is formed.

- The groynes are placed at equal distance with an optimum spacing of four metres at inner bank and seven meters at the outer bank so that the groins can offer maximum protection to the river bank between them.

- It is observed that Optimum condition for river bank protection is found when the groynes are aligned as both attracting and deflecting.

[1] Balen, W., Uijttewaal, W. S., and Blanckaert, K. (2009). "Large-eddy simulation of a mildly curved open-channel flow" Journal of Fluid Mechanics, 630(1), pp. 413-442.

[2] McCoy, A., Constantinescu, G. "Numerical Investigation of Flow Hydrodynamics in a Channel with a Series of Groynes".

[3] Uijttewaal, W.S.J. "Effects of Groyne Layout on the Flow in Groyne Fields: Laboratory Experiments", Journal of Hydraulic Engineering (2005).

[4] Rajaratnam, N., Nwachukwu, B. A. "Flow near Groin-Like Structures", Journal of Hydraulic Engineering, Vol. 109, Issue 3, January 1983.

[5] Schmidt, J. C. "Flume Simulation of Recirculating Flow and Sedimentation", Water Resource Research (1993).

[6] Burele, S., Gupta, I.D., Sharma, N. "Experimental Study on Performance of Spurs", ISH Journal of Hydraulic Engineering, September 2012.

REFERENCES