Simulation of flow pattern around series of groynes with different orientations in meandering channels

K P Indulekha1,2, P K Jayasree and S Sachin
Department of Civil Engineering, College of Engineering Trivandrum
2indulekhakp@gmail.com

Abstract. The mechanism of flow and turbulence around groynes in a meandering channel is very complex phenomena. To simulate the flow pattern around groynes in meanders a 3D numerical model was built with groynes of different inclinations by using ANSYS Fluent. The developed model was verified by the results of experimental study in meanders. Pressure, velocity, stream lines and turbulent kinetic energy for different groyne angles such as 15°, 30°, 45°, 90°, 120° and 150° were compared and analyzed to find the ideal angle configuration for different functions. Streamlines profiles were studied to understand more about the formation of scours and vortices around the groyne. The study of streamline profiles helped in the study of vortex and scours. The most dynamic variation in pressure and turbulent kinetic energy within the contours were found for groynes with 90° orientation. Large numbers of vortices between the groynes were observed for groynes with 15° and 90° orientation and these vortices were observed between the second and third groynes. For groynes in the meandering, the ideal angle for controlling the velocity was found out to be 45°, as it showed very small number of vortices in the groyne field. The results of the study showed that numerical modelling could be used efficiently for the study of flow and turbulence around groynes in meanders.

1. Introduction
Flow pattern around series of groynes in meanders is a complex phenomena in natural rivers. Therefore a better understanding of turbulent flow is needed to analyse sediment transport in meanders. To analyse scouring around such structures, the mean flow and turbulence distributions around these structures has to be analysed properly. Groynes or Spur in rivers and streams are used to prevent bank erosion, and to keep the main channel navigable. The complexity of flow around series of groynes, increases with the development of the scour hole in meanders. Outer banks of river bends are usually associated by scouring. As a result lateral migration of channel may take place. The spur dike may be used in a channel bend to control the bank scour and its lateral migration[1]. Flow behaviour downstream of the spur dike is also very complicated. It is mainly a combination of vertical up flow vortices, horizontal vortices and reverse flow in the spur dike zone [2]. A series of groynes serve to confine the flow in a narrow main channel to keep it away from erodible banks. The most important considerations involved in groyne design in meanders are plan view shape, length, spacing and orientation to the flow. Even though many experimental and numerical studies have examined the mean flow and turbulence structures around series of groynes in straight channels, only few studies are reported on flow around groynes in meanders. Some experimental studies related to the measurement of turbulence quantities for flow around bends exist [3,2,4,5]. Flow pattern around single spur in channel bends were analysed by various researchers [6,7,8]. The simulated down-stream and cross-
stream mean flow property as well as turbulent intensities in shear layer of spurs protruding from the bank of a meandering-like laboratory flume with smooth rigid bed was found to be in good agreement with the experimental results [6]. The horseshoe vortex at the base of the straight groyne has found more compact and strong rotational momentum and lasts a longer distance downstream compared with T-shape groynes [3]. In the literatures reviewed, the flow patterns due to a single groyne were analysed for the rigid bed material of the channels. The objective of the present study is to analyse the flow pattern around series of groynes in meanders for erodible bed under different orientation angles and to identify a suitable orientation of groynes in river meanders.

2. Numerical modelling

Numerical modelling involves the solution of Navier Stokes equation, which is based on the assumption of conservation of mass and momentum in a moving fluid. The continuity equation for conservation of mass, equation for conservation of momentum can be expressed as:

\[ \nabla \cdot \mathbf{U} = 0 \]  
\[ \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \mathbf{U} \mathbf{U} = -\frac{1}{\rho} \Delta P + \nu \nabla^2 \mathbf{U} + f_b \]  
Where \( \mathbf{U} \) is velocity, \( P \) is pressure and \( f_b \) is body forces

The standard k-\( \omega \) model used for numerical modelling is an empirical model based on model transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (\( \omega \)). This model underwent numerous modifications to improve its accuracy and as a result the transport equations formulated as follows.

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k \mathbf{U}_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - y_k + S_k \]  
\[ \frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega \mathbf{U}_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - y_\omega + S_\omega \]  
\( G_k \) represents the generation of turbulence kinetic energy due to mean velocity gradients. \( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of k and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of k and \( \omega \) due to turbulence. \( S_k \) and \( S_\omega \) are user-defined source terms.

3. Physical Experiments

Experiments were conducted in a meandering flume featuring a meandering portion of a Vamanapuram river in Kerala at the PG Hydraulic Laboratory of College of Engineering, Trivandrum. The curved portion of the river was scaled down to a horizontal scale ratio of 1:60 and vertical scale of 1:20 are shown in Figure1(a). The flume consists of three consecutive and opposite bends with angles 67°, 134°, and 67°(Bend 3). The central angle of 134° represents typical meander geometry. The physical model has a cross section with base width 50mm and top width 130mm and the side slope of the model was 39°. The flume is filled with river sand collected from the banks of river. The sand has an effective size of 0.5mm and mean size of 0.88m. The sediment settling basin was designed to have a drop of 600 mm to the laboratory floor to allow the sediment to settle after being washed out of the channel. A calibrated V-notch fixed at the downstream of the flume was used for measuring the discharge. In order to ensure that all the sediment would not be eroded away, a sediment layer of 100 mm was used in the channel. Considering extreme conditions of the Vamanapuram river basin, discharge was fixed as 27 lps for all the experiments. The experiment was conducted at flow velocity of 0.41m/s with series of groynes at spacing by length (S/L) ratio of 2.5. The velocities at different sections on the physical model were observed using 16MHz Acoustic Doppler Velocimeter. In all cases, the Froude number of 0.35 was maintained which is small enough to ensure subcritical flow and the Reynolds number was high enough to ensure fully developed
turbulent flow. A movable bed river model was setup in the numerical model to represent a curved reach as in the prototype. For numerical modelling Ansys Fluent was used, which applied computer simulation methods to analyses fundamental principles of computational fluid dynamics (CFD) such as the conservation of mass, momentum and energy. The flume geometry created in ANSYS FLUENT is shown in Figure 1 (b). Five cross sections X1, X2, X3, X4 and X5 respectively as shown in Figure 1 (c) were considered around the groyne field for detailed analysis in numerical model. The algorithm for the numerical modelling used was SIMPLE method and Volume of fluid principle was followed and it was inputted in the phase window.

4. Results and discussions
In the numerical modelling a series of groynes were arranged on the outer bank of the central curve of the meandering flume. Groynes were arranged for different angles of 15°, 30°, 45°, 90°, 120° and 150° for an S/L ratio of 2.5. Pressure, velocity, stream lines and Turbulent Kinetic Energy (TKE) of different orientations were simulated in Ansys Fluent. The result obtained from the experimental model was compared with the velocities from the numerical model as shown in Figure 2. The results obtained from the numerical model shows a good agreement with the experimental results.

4.1. Simulated Profiles for different angles

4.1.1. Pressure. Figure 3 shows the pressure contours for different orientation angle of 15°, 30°, 45°, 90°, 120° and 150°. As shown in Figure 3, the maximum pressure was observed in the 150° orientation (Figure 3(f)) and major pressure variations was observed in the third and fourth groyne and the angle which experienced maximum pressure variation was the 90° setup and the variations in pressure was observed around 2nd, 5th and 6th groynes. For section X2, the pressure gets reduced by 30%. Then it remains almost same till X4 then it increases as it reaches the last section for angle of 45°.
4.1.2. Velocity. The velocity profiles computed from numerical model is shown in Figure 4. The velocity was very low in the groyne field for 15° and 150° configuration, but nearer to the groyne field a dynamic fluctuation in the velocity was observed. For 30° and 45°, velocity increases as it entered the groyne field and then decreases after the groyne field. But for the 90° and 120° orientations, the velocity increase throughout the field. The groynes of 15° and 150° orientation experienced the most number of vortices. In the 30° and 45° orientation, the velocity decreases when it enters the groyne field and 90° configurations showed the maximum velocity in the profiles.

4.1.3. Turbulent Kinetic Energy. The maximum Turbulent Kinetic Energy was observed for the groyne with 90° orientation. For the groyne orientation of 15° and 150°, similar patterns were observed. For groynes arranged at angles of 45° and 90° shows less variation in TKE when compared to other angles. Most variations were focused around the third groyne for all angles as shown in Figure 5.

4.1.4. Stream Lines. The streamlines generated for different orientation of groynes are shown in Figure 6. It is seen that strong vortices were developed in the groyne fields for angles 15°, 90° and 120° degree angles. Vortices were also developed in between first and second groyne for orientation angle of 90° and 120° degree. For 15° the flow lines emerges into groyne region that will cause the erosion of sediments in the groyne fields.

4.1.5 Variations along sections. Pressure and Turbulent kinetic Energy were maximum at section X1 and then it gradually decreases by 10% to 20% when it reaches the last section X5. In the case of 45° orientation, velocity was maximum at sections X3 and X5 and it reduces by 30% at section X1. When comparing the angles with change in velocity, 15° and 150° orientation showed very less changes. For the 30° and 45° orientations, the velocity initially increases when it enters the groyne field and then this velocity is reduced very much due to the effect of groynes. But for groynes with 90° and 120° orientation the velocity increases after the groyne field. When the pressure and Turbulent Kinetic Energy was analyzed, groynes with 90° orientation has shown the most dynamic variation in properties like pressure and turbulent kinetic energy within the contours. The 15° and 150° orientation showed the very less turbulent kinetic energy and it was very similar to that of the condition without groynes. The Pressure contours showed that, as the angle increased the pressure was also increased. For 15° orientation there was only a small pressure variation at the second and third groynes. The study of streamline profiles helped in the study of vortex and scours. Groynes with 15° and 90° orientation have shown the most number of vortices within the profiles. These vortices are more concentrated within the second and third groyne. For 45° orientation, velocity gets increased by 50% at section X2 and then the velocity gets reduced by almost 60% at last section X5. The velocity increases by 30% at section X2 and then it remains almost the same up to section X4 after that the velocity reduces by almost 10%.

![Figure 2. Comparison of velocity form Numerical Model with the Experimental Model](image-url)
Figure 3. Pressure contours for different orientation of groynes (a) $15^0$, (b) $30^0$, (c) $45^0$, (d) $90^0$, (e) $120^0$, (f) $150^0$

Figure 4. Velocity contours for different orientation of groynes (a) $15^0$, (b) $30^0$, (c) $45^0$, (d) $90^0$, (e) $120^0$, (f) $150^0$

Figure 5. TKE contours for different orientation of groynes (a) $15^0$, (b) $30^0$, (c) $45^0$, (d) $90^0$, (e) $120^0$, (f) $150^0$

Figure 6. Streamlines for different orientation of groynes (a) $15^0$, (b) $30^0$, (c) $45^0$, (d) $90^0$, (e) $120^0$, (f) $150^0$
4.2. Variation of parameters for 45° angle

Figure 7 shows the change in Turbulent Kinetic Energy along each section. The maximum TKE was observed at first section X1 and it is reduced by 33% to 50% on the last section. If a series of groynes are arranged in meanders TKE can be considerably reduced. The maximum Turbulent Kinetic Energy was at the tip of the groyne at first section X1 and then gets reduced by 30 % when it reaches X4. Then it becomes almost a constant throughout the flume. Figure 8 shows the variation of velocity at different sections. It is seen that for all the sections, except for X1 the velocity profile is almost same for 45° angle. And the velocity at these sections are lesser than the approach velocity of 0.41 m/s. 45° angle is found suitable for velocity reduction in meanders.

![Variation of TKE at different sections for 45° angle](image)

**Figure 7.** Variation of TKE at different sections for 45° angle

![Variation of Velocity at different sections for 45° angle](image)

**Figure 8.** Variation of Velocity at different for 45° angle

5. Conclusion

Numerical modelling for study of flow pattern around groynes was found to be very effective, the results obtained from the experimental model was compared with that of the numerical model and the results obtained showed good agreement with the experimental values. The study of profile contours helped in understanding more about interaction between the flow and groynes. Even though groynes aligned at 15° and 150° with bank showed the least effect on the velocity, it cannot used as a river protection work as it does not deflect the flow away from the banks. For groynes with 90° orientation showed the most dynamic variation in properties like pressure and turbulent kinetic energy within the
Contours. The study of streamline profiles helped in the study of vortex and scours. Groynes with 15° and 90° orientation showed the most number of vortices within the profiles these vortices all happened within the second and third groyne. Most of the maximum values of pressure, velocity and turbulent kinetic energy were experienced in the tip of the groyne. The ideal angle for controlling the velocity was found out to be 45°, as 30 to 50% of reduction in the velocity is observed when the flow reached the downstream.

Acknowledgement
The authors are thankful to the Kerala State Council for Science, Technology and Environment, Kerala, India for providing financial assistance for research work under Engineering Technology Programme (ETP).

References
[1] Ghodsian, M., Vaghefi, M. (2009), “Experimental study on scour and flow field in a scour hole around a T-shape spur dike in a 90° bend”, International Journal of Sediment Research, 24(2), 145-150, https://doi.org/10.1016/S1001-6279(09)60022-6
[2] Fazli m, Ghodsian M, & Neyshabouri, S. A. A. S., (2008), “Scour and flow field around a spur dike in a 90° bend”, International Journal of Sediment Research, 23(1), 56-68.
[3] Safarzadeh, A., Neyshabouri, S, A, A, S.,and Zarrati, A, R.,(2016), “Experimental Investigation on 3D Turbulent Flow around Straight and T-Shaped Groynes in a Flat Bed Channel”, ASCE Journal of Hydraulic Engineering, 142(8),
[4] Vaghefi, M., Ghodsian, M., Akbari, M., (2017), “Experimental Investigation on 3D Flow around a Single T-Shaped Spur Dike in a Bend”, Periodica Polytechnica Civil Engineering, 61(3), pp. 462-470., https://doi.org/10.3311/PPci.7999
[5] Zhang H., Nakagawa H., Kawake K., & Baba Y.(2009), Experiment and simulation of turbulent flow in local scour around a spur dyke, International Journal of Sediment Research, 24 (1) (2009) 33–45, https://doi.org/10.1016/S1001-6279(09)60014-7.
[6] Giri, S., Shimizu, Y., & Surajate, B. (2004). “Laboratory measurement and numerical simulation of flow and turbulence in a meandering-like flume with spurs”. Flow Measurement and Instrumentation, 15(5-6), 301–309. doi:10.1016/j.flowmeasinst.2004.05.002
[7] Yazdi,J., Sarkardeh,H., Md. Azamathulla,H. & Ab. Ghani A., (2010),“3D simulation of flow around a single spur dike with free-surface flow”, International Journal of. River Basin Management, 8:1, 55-62, doi: 10.1080/15715121003715107
[8] Nagata, N., Hosoda, T., Nakato, T., Muramoto, Y. (2005). "Three-dimensional numerical model for flow and bed deformation around river hydraulic structures." Journal of Hydraulic Engineering. 131(12), pp. 1074-1087. https://doi.org/10.1061/(ASCE)0733-9429(2005)131:12(1074)
[9] Bonakdari, H., Baghalian, S., Nazari,F., & Fazli,M., (2011) “Numerical Analysis and Prediction of the Velocity Field in Curved Open Channel Using Artificial Neural Network and Genetic Algorithm”, Engineering Applications of Computational Fluid Mechanics, 5:3, 384-396, doi: 10.1080/19942060.2011.11015380