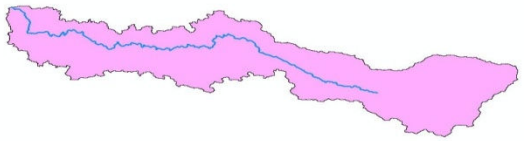
Hydrodynamic Modelling of River Flow

Navneet Kumar
Department of Civil Engineering, IIT Delhi

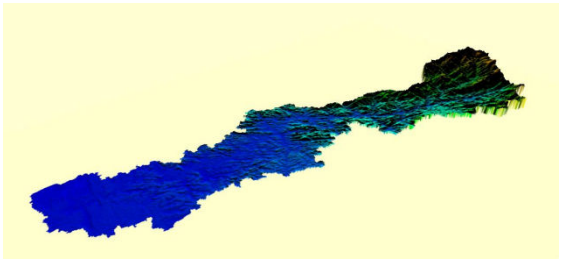
The hydraulic characteristics of natural river flood plains are not well understood at present. This is due to the problems encountered in monitoring spatially distributed patterns of flow depths, velocity, turbulence characteristics etc. For designing the flood protection strategies, it is very important for river engineers to accurately predict water levels that may be expected due to any flood discharge. One of the consequences resulting from the more recently recognized hazards of climate change is the potential to increase the levels and occurrence of flooding worldwide. All this requires research on natural river flows. Meandering channel flows being highly complicated are a matter of recent and continued research. In fact, the need for a computational fluid dynamics (CFD) package for river flow hydrodynamics problems is highly increasing. In the present work computational fluid dynamics (CFD) simulations are carried out of Achankovil River in kerala.

The Achankovil Aar is a river in Kerala, India, formed towards the southern tip of the peninsula from the streams of the Rishimala River, Pasukidamettu River, and the Ramakkalteri River. This river enriches the Pathanamthitta district of Kerala state. It joins with the Pamba River at Veeyapuram, in the Alappuzha district of Kerala in South India.

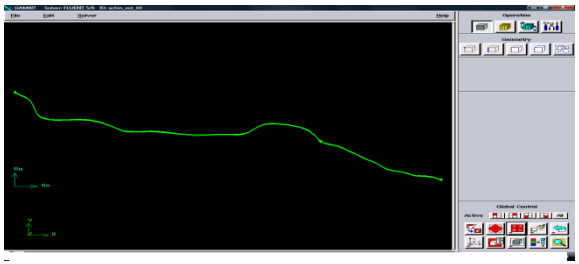
Achankovil Basin in kerala



Achankovil Basin 3D Geometry



Achankovil 3D Geometry in Gambit



The computational modeling of three dimensional flows of Achankovil River has been performed in this research work. The flow calculations are performed by solving 3D steady state continuity and Reynolds averaged Navier-Stokes equations. The latter are based on the solution of the complete set of Reynolds-Averaged Navier-Stokes (RANS) equations coupled to the Volume of Fluid (VOF) method. The turbulence closure is approximated with standard-turbulence model. The model equations are solved numerically with a general purpose software package.

GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications. **Gambit is a mesh generator, and Fluent a flow solver**.

The volume of fluid (VOF) turbulence model is applied to obtain characteristics of three-dimensional open channel flows involving free surfaces. In particular, the VOF model is used to determine the pressure head distributions, velocity distributions, and water surface profiles for the free overfall in open channel. The predictions of the proposed model are validated using existing experimental data.

References

Biscarini, C., Di Francesco, S., & Manciola, P. (2010). CFD modelling approach for dam break flow studies. *Hydrology and Earth System Sciences*, *14*(4), 705-718. doi:10.5194/hess-14-705-2010.

Kennedy, M. G., Ahlfeld, D. P., Schmidt, D. P., & Tobiason, J. E. (2006). Three-Dimensional Modeling for Estimation of Hydraulic Retention Time in a Reservoir. *Journal of Environmental Engineering*, *132*(9), 976. doi:10.1061/(ASCE)0733-9372(2006)132:9(976).
(14383).