International Review of Applied Engineering Research.
ISSN 2248-9967 Volume 4, Number 4 (2014), pp. 337-342

http://www.ripublication.com/iraer.htm

CFD Analysis in the Field of River Engineering: A Review

[1]R. Karthik, [2]U. Kumar and [3]K.M. Pandey

[1]Department of Civil Engineering, NIT Silchar, Assam, India.
[2]Department of Civil Engineering, NIT Silchar,, Assam, India.
[3]Department of Mechanical Engineering, NIT Silchar, Assam, India.

Abstract

This review paper reports some further research scope on computational fluid dynamic (CFD) on the river related problem like flooding, river migration and the effect of pre-scour hole around the hydraulic structure, Modelling of alluvial roughness in complex and dynamic domain, simulation of multiphase flows with a free boundary including possible phase transitions in flooding. Here discussed few numerical models like Finite volume method, Finite element method, Reynolds stress model, Reynolds averaged naveir-stokes (RANS), Detached eddy simulations (DES) and type of grid formation, boundary condition and different kind of work done so far by researchers in this field, like the transient bed changes and flow structure, behaviour in a narrow sharp 90◦ and a 180◦ channel bend were successfully modelled. While flowing the sharp curvature of river bend, the flow properties like velocity, kinetic energy, turbulence, etc is changed and it will initiate the process of erosion and deposition on the channel.

Keywords: CFD, river engineering, erosion, bed migration.

1. Introduction

In the field of river engineering, there are many complex process are occurring inside the river, during the last century most of the research in the field of river engineering are carried by field observation and laboratory experimental. But in this method the amount of information are very limited and they are subjected to human errors. Some complex flow domain could not take the measurement of some flow properties and could not make the model for laboratory experiment also, and it is very expensive, time consuming. Alternatively in the computational analysis some mathematical models are using, in this model consisting several differential and algebraic equation and which is make possible to predict. During the last two decades the computing power of digital computers and efficiency of commercial CFD codes are tremendously increasing, and it's very less time consumption, economic, we can analysis a complex flow domain with predict all the flow properties which is not possible in lab and field experiment.

2. Litreature Review

A natural meandering stream is one of the most complex water flow situations encountered in real life, in the river engineering predicting the flow behaviour on the sharp curvature of the river is essential., Because in the sharp bend helical flows occurs and it is the major cause of erosion and deposition problem. The outer bank of the river is called concave and the inner bank is called convex. The main reason of helical flow is the difference of centre of gravity of upper and lower layer of flow in meandering river. While flow through sharp curvature of the river, the top layer of water go to outer bank and bottom layer will go to inner bank this process is called as flow separation, so the velocity of flow near the outer bank is higher compared to inner bank. In the linear theory of Ikeda et al. (1981) the migration rate is assumed to be proportional to the near-bank water flow velocities. So the flow will erode the bank material from outer bank and the fine particle flowing through water settle down in the inner bank because of velocity. The water level of outer bank is always high when compared to inner bank of the bend of meandering river, so this will initiate to form the counter-rotating secondary flow. In the 90° and 180° river bend analysis showing, the flow velocity is maximum on upstream on outer bank, the contour of constant lateral velocity shows the water movement towards the outer bank with zero contours near bend. And the turbulent production by kinetic energy tends for lateral direction being large near the outer region of bend and decrease towards inner bend and the total pressure is uniform before the bend and continuously decreasing after the bend.

Flooding is the recurring natural event and it is severely impacting the human lives and damaging the properties and this depends upon the magnitude of flooding. India and Bangladesh is the most flood prone nation on earth. Nowadays water is an essential for generation of electricity and agricultural purpose so we need to construct the dam to storage the water along the river but unfortunately dam breaking will lead to flooding. Were successfully modelled the effect of dam break, fully three dimensional simulation by using of CFD models Reynolds-Averaged Navier stokes (RANS) equation coupled to the volume of fluid method. (Biscrini, Francesco and Manciola, 2010), and the VRV is developed the flood hazard mapping by using of AVL CFD (SWIFT) package. It is more accurate flood risk assessment and it will help to calculate the financial risk, and gives a more accurate warning system. The swift CFD codes are working on the basis of Finite Volume approach and the conservation equation of mass, momentum and energy for multi phase flow. (Mohamed et al, 2002)

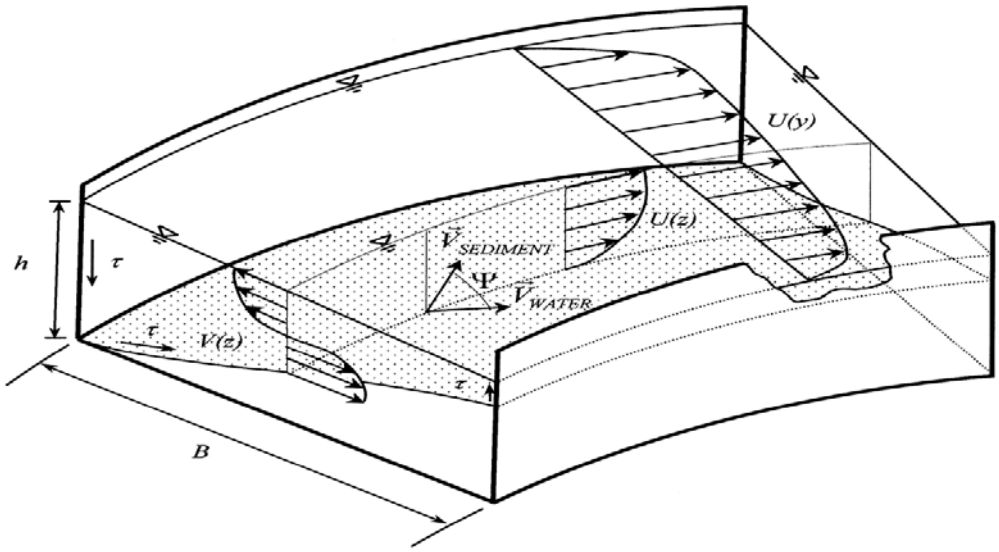

Fig. 1: Sketch of flow and sediment transport direction in a channel bend, this photograph appears courtesy of [Nils Ruther, 2006].

One of the prime reason for failure of hydraulics structure is scour hole around the pier of the bridges. The statics showing around the world 8o percentage of bridge failure were the cause of piers washed –out, so designer need to consider while designing and lot of researches are carried out to predict the rate of erosion occurring the scour hole by using of various CFD models like large-eddies simulation (LES) and RANS, Sometimes comparing result of both LES and RANS (Pasiok, and Szydlo, 2010).

The finite volume method is one of the CFD model, it is relatively easy to understand and use, its governing the partial differential equation and solve over a control volumes, especially in the form of first and second order approximation on the structure meshes. And it is easily formulated to allow the unstructured mesh. The FEM discretisation guarantees the conservation of fluxes through a particular control volume.

The Finite Element method is one of the numerical techniques to find the approximate solution for the differential equation of boundary value problems. For variational method it's used to minimized the functional errors and similarly produce the stable solution. The FEM formulation requires a special care of ensuring a conservative solution. However, FEM can require more memory and has taken more time to produce the solution than the FVM.

The Reynolds Averaged Navier-stocks model (RANS) for time-averaged equation of motion for fluid flow, In this RANS model having two types, one is Boussinesq hypothesis and another is Reynolds stress model (RSM**).** The RANS equations are primarily used for describe the turbulent flows and this equation can be used with approximation based on the knowledge of turbulence flow properties to give a time-averaged approximate solution to the navier-stocks equation.

Detached eddy simulation (DES) is modified form of RANS model and near the solid boundaries the turbulent length is less than maximum size of grid dimension in that case it will assigned the simulation by RANS mode, and the turbulent scale length high than grid dimension this problem solved by using of LES mode, both model is very useful for predict the rate of erosion and scour on near the bed and bank of the river.

In the entire flow domain will dividing small rectangle or triangle cell, because each and every cells the analysis will carry and produce more stable solution. The group of cell called as boundary zone, in CFD many varieties of grids are available depend upon the size, shape, structure and unstructured.

The boundary condition is the numerical values, which is used to describe the flow domain, there is two types of boundary condition are there like cell centre, cell face. In all sides of the flow domain have to consider the boundary condition like inflow, outflow, wall surface condition, velocity, turbulence, pressure.

3. Work Done so Far by Previous Reaserchers

Recent studies have used 3D models to simulate the flow behaviour around the bends and the mechanisms of secondary flow structure (Morvan et al 2001, Booji 2003, Blanckaert and De Vriend 2004, Khosronejad et al 2007, Zeng et al 2008, Han et al 2011)

Several 3D CFD models based on navier-stokes equation already used to investigate the river engineering problems like flow over weir, landslide generate waves, through bridge piers and dam breaks (G omez-Gesteria and Dalrymple 2004, Nagata 2005, Quecedo et al 2005, Ling et al 2007, Mohammedi 2008, Biscarini 2009)

Khosronejad et al (2007) developed a 3D morphodynamic model and simulate the bed evolution in 90° and 135° channel bends, and Wu et al (2000) used a 3D numerical model to simulate the flow and sediment transport in a 180° bend

4. Present Research Scope

In the field of river engineering the flooding is one of the prime issue, and the main reason of flooding is the rainfall intensity of river catchment area, so the wetting and drying process of river catchment area is modelled, but the accuracy of the simulated results is strongly depended on the stability performance of the numerical model. The present model showed some limitations and drawbacks especially when dealing with unstructured grids in combination with the modelling of wetting and drying processes. It is a matter of great interest to solve these problems in future investigation.

Modelling of alluvial roughness is extremely complex and dynamic. More research has to be done in the field of measuring the initiation growth, movement and the influence of bed forms on the flow field. Today such numerical implementation is limited and still under developed.

In the river bed migration is the continues process on the meandering river, so to control the river migration by altering the local flow structure such as upstream flow depth, downstream velocity and bed shear stress distribution, they used to construct the rock weirs. But further research is recommended for rock weirs in a sinuous, mobile-bed channel with symmetric and non-symmetric weir configurations. By altering the structure geometry of U weirs the maximum increase in channel velocity ranged from 1.24 to 4.04 times the reference velocity in the channel with no structure present (Christopher,2011). Investigations of variations in channel characteristics, non-symmetric structure geometries, and mobile bed simulations would provide valuable data for addition to the current data set. Investigation of these conditions should include a range of approach conditions, structure configurations, and channel geometry that would alter the resulting local hydraulics, scour development, and potential stability of the structure. Such research could be conducted in the laboratory setting as well as through the use of a mobile bed numerical model. Incorporating a testing scheme that includes both physical and numerical modelling components is recommended.

In river engineering another important problem is to identify the pre-scour hole around the hydraulic structure, in the structure typically involves a pre-excavated scour hole downstream of the structure crest. However, design guidelines related to the location and size of the scour hole and its effects on the local hydraulics of the weir does not exist. Being able to understand how variations in the channel geometry of the scour holes downstream from rock weirs affect the local hydraulics like velocity and bed shear stress and resulting scour development and sediment continuity is important in the design and sustainability of the structure. So the effect of pre-scour holes could to be investigated through additional numerical modelling and through physical model testing in the laboratory. Simulation of multiphase flows with a free boundary including possible phase of transitions in a flood is still research topic.

5. Conclusions

In this review paper we have presented about history and development of computational fluid dynamic CFD, commercially available computer codes based on CFD models and strategy of the CFD are discussed elaborately. The recent developments in computer software has advanced the use of computational fluid dynamics CFD in analyzing the character and behaviour of flow, the field application of CFD particularly in this river engineering related issue like flooding, river migration, scour hole on the hydraulic structure and further research scope on CFD modelling like effect of pre-scour hole and a rock weir in sinuous, mobile bed channel, modelling of alluvial roughness in complex and dynamic domain. This paper discuss the solution of CFD simulation on sharp curvature of the meandering river bend and how the flow properties will changed depend on effect of curvature on bend and the CFD modelling is able to depict the physical quantities such as velocities and pressures of the fluid, which is very useful information for future engineering design, etc.

References

[1] A. Khosronejad, C.D. Rennie, S.A.A. Salehi Neyshabouri and R.D. Townsend (2007), 3D Numerical Modellinfg of Flow ad Sediment Transport in Laboratory Channel Bends, *J.Hydraul. Eng.,* **133,** 10, pp.1123.

[2] A.S. Ramamurthy, S.S. Hnaz and P.M. Biron (2013), three-Dimensional Simulation Parameters for 90° Open Channel Bend Flows, *journal of Computing in Civil Engineering,* ASCE, PP.282-291.

[3] Congfang Ai, Sheng jina, and Yan Xing (2013), Influence of Suspended load on 3D Numerical Simulation of Flow and Bed Evolution in a Meandering Channel Bend, *Journal of Hydraulic Engineering, ASCE*, pp.450.

[4] C. Biscarini, S. Di Francesco and P. Manciola (2010), CFD modelling approach for dam break flow studies, *Hydrol. Earth Sys.,* **14**, pp.705-718.

[5] Christopher Lee Holmquist-Jhonson (2011), Numerical Analysis of River Spanning Rock U-Weirs: Effects of Structure Geometry on Local Hydraulics, *thesis for the Doctor of Philosophy,* Colorado State University.

[6] J. Zeng, G. Constantinescu and L. Webel (2008), A 3D Non-hydrostatic Model to Predict flow and Sediment Transport in Loosed Bed Channel Bends, *J. Hydraul. Res.,* **46**, 3, pp.356-372.

[7] K. Blanckeart and H.J. De Vriend (2004), Secondary Flow In Sharp Open channel Bends, *J.Fluid Mech.,* **498**, 1, pp.353-380.

[8] M. Mohammadi (2008), Boundary Shear Stress Around Bridge Piers, *J. for Applied Science***, 5,** 11, pp.1547-1551.

[9] Mohammed Gouda, Dr. Konard Karner, Dr.Reinhard Tatschl (2002), Dam Flooding Simulation Using Advanced CFD Methods, *WCCMV Fifth world congress on Computational Mechanics*, Austria, pp.2.

[10] M. Gomez-Gesteria, and R.A. Dalrymple (2004), Using a Three Dimensional Smoothed Particle Hydrodynamics method for Wave impact on Tall Structure*, J. Waterway, Port, Costel and ocean Eng.*, **130**,pp.63-69.

[11] M. Quecedo, M.Poster, M.I. Herreros, J.A. Fernandez merodo, Q. Zhan (2005), Comparison of two mathematical models for Solving the Dam Break Problems Using the FEM Method, *Comput. Methode Appl*, **194,** (36-38), pp.3984-4005.

[12] N. Nagata, T. Hosoda, T. Nakota, Y. Muramoto (2005), Three Dimensional Numerical Model for Flow and Bed Deformation around river hydraulics, *J.Hydraulics Eng, ASCE*, 131(12), pp.1074-1087.

[13] Nils Ruther (2006), Computational fluid dynamics in fluvial sedimentation engineering, Thesis for the degree of doctor philosophiae, Norwegian University of Science and Technology, pp. 10.

[14] R. Pasiok and E. Stilger-Szydlo (2010), Sediment particles and turbulent flow Simulation around bridge piers, *Archives of Civil and Mechanical Engineering,* Vol. X, NO.**2**.

[15] S. Ikeda, G. Parker and K. Sawai (1981), Bend theory of river meanders, linear development, *J. Fluid Mec*hanics, **112**, pp.363-377.