ANALYSIS OF BOUNDARY SHEAR STRESS IN MEANDERING CHANNEL USING ANSYS FLUENT

Dissertation submitted in partial fulfilment of the requirement for the

Award of degree of

MASTER OF TECHNOLOGY

IN

HYDRAULICS AND WATER RESOURCES ENGINEERING

BY

YUVARAJ
(2K15/HFE/16)

Under The Guidance of

Mr. S ANBU KUMAR

Associate Professor
Department of Civil Engineering
Delhi Technological University
Delhi

DELHI TECHNOLOGICAL UNIVERSITY (FORMERLY DELHI COLLEGE OF ENGINEERING) DELHI-110042

JULY 2017

<u>CANDIDATES'S DECLARATION</u>

I do hereby certify that the work presented is the report entitled **"ANALYSIS OF BOUNDARY SHEAR STRESS IN MEANDERING CHANNEL USING ANSYS FLUENT**" in the partial fulfilment of the requirements for the award of the degree of "Master of Technology" in Hydraulics & Water Resources Engineering submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of my own work carried out from January 2017 to July 2017 under the supervision of Mr. S ANBU KUMAR (Associate Professor), Department of Civil engineering.

I have not submitted the matter embodied in the report for the award of any other degree or diploma.

Date: 18/07/2017

Yuvaraj
2K15/HFE/16

<u>CERTIFICATE</u>

This is to certify that above statement made by the candidate is correct to best of my knowledge.

Mr. S ANBU KUMAR
(Associate Professor)
Department of Civil Engineering
Delhi Technological University

ACKNOWLEDGEMENT

I take this opportunity to express my profound gratitude and deep regards to Mr. S ANBU KUMAR (Associate Professor, Civil Engineering Department, DTU) for his exemplary guidance, monitoring and constant encouragement throughout the course for this project work. The blessing, help and guidance given by him from time to time shall carry me a long way in life on which I am going to embark.

I would also like to thank Prof. Nirendra Dev (Head of Department, Civil Engineering Department, DTU) for extending his support and Guidance.

Professors and faculties of the department of Civil Engineering, DTU, have always extended their full co-operation and help. They have been kind enough to give their opinions on the project matter; I am deeply obliged to them. They have been a source of encouragement and have continuously been supporting me with their knowledge base, during study. Several of well wishers extended their help to me directly or indirectly and we grateful to all of them without whom it would have been impossible for me to carry on my work.

ABSTRACT

Research has been carried out on meandering channels to find various aspects of boundary shear stress. But no methodical work has been done to find the variation of boundary shear stress along a path of meandering channel. In this particular research work, deviation of shear stress distribution along the depth and width of the meandering channel has been systematically analysed at different cross-sections along a path of sinuous meandering channel of 120° angle. The path of the meandering channel is considered from one apex of the bend to the next apex of the bend which changes its direction at the crossover. Bend apex represents the point of maximum curvature and crossover is the section where the sinuous channel changes its sign.

The analysis has been performed at 13 different different sections along the path of the meandering channel, that is from one apex of the bend to another. Research has been done to analyse the variation in boundary shear stress along the wetted perimeter on the channel bed with different discharges to depict the flow characteristics of meandering channel.

The research has been done through numerical modelling technique using ANSYS FLUENT which simulates model to solve equations.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1. INTRODUCTION

1.1 Channel

A channel is a wide strait or waterway between two land masses that lie close to each other. It can also be the deepest part of a waterway or a narrow body of water that connects two larger bodies of water.

Channel analysis is necessary in order to assess

1. Potential flooding caused by changes in water surface profile
2. Disturbance of river system upstream or downstream of the highway right of way
3. Changes in lateral flow distribution
4. Changes in velocity or direction of flow
5. Need for conveyance and disposal of excess runoff
6. Need for channel lining to prevent erosion

1.2 Meandering River

Meandering river tends to form on low topographic gradients and when and when there is high proportion of suspended sediment relative to bed load transport. One of the key aspects of meandering river is that the meanders tend to to grow with time because as a water enters a bend, the flow on the inside of the bend tends to be much slower than the flow on the outside. The water coming through the outside has to swing around much more quickly than on the inside. From general relationship and thinking about the Reynolds number which depends on the flow speed one can predict the erosion on the outside of the bend and deposition on the inside of the bend that means through time the bends tends to grow and become larger because the outside of the bend is eroding and the inside is depositing. Large bends called meanders has different associated landforms as shown in figure 1.1. Figure shows the meandering, graded stream, oxbow lake, meandering scars, meander scrolls, Yazoo stream, bluffs, undercut bank, back swamp, floodplain and natural levee.

Meander scroll: Occasionally meander scarp is actually a geological features formed by remnants of the meandering channel. They are formed during the formation of oxbow lake.

Meander scars: They are caused by varying velocities in the river channel due to higher velocities current on the outer side of the river channel through bend.

Yazoo Stream: Yazoo stream or tributary is the smaller channel that flows parallel to the larger channel on the flood plain of the larger stream. It does not get to the main stream because there is a kind of levee which has the higher elevation right alongside the river and they can extend up parallel to the river.

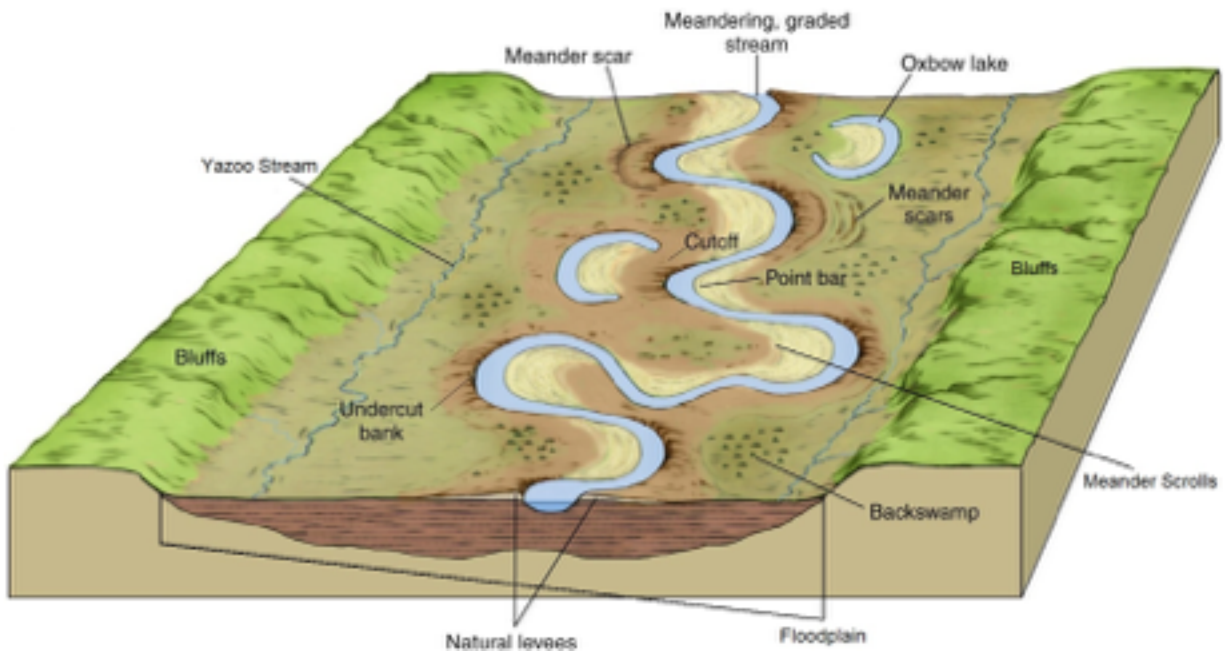

Fig 1.1. Meandering river and its associated landforms.

Two channels close together erode and deposited and will get closer to each other. Eventually often in a flood the barrier between the two channels get eroded away and the channel straightens out and begins to flow, which is generally called cut off. Water starts flowing through the cutoff and the flow speed slows down through meander end up depositing sediments along the side of the channel which left with an empty channel that often forms a oxbow lake.

There is a variation in a water depth across the profile of the meandering channel as we go from one side to the other. The Reynolds number depends both on flow speed and water depth, the highest water depth and the fastest flow speed are on the outside of the bend. So, it has the highest Reynolds number. That means one can transport the sediments and it is often being transported as dunes with trough cross stratification and the courses grains. The inside of the bend where the flow speed is lower and the water depth is lower the Reynolds number is much lower in that case one can only transport much finer sediments and with lower flow speed it is ripple cross laminated. Meanders are typical landforms found in this stage of the river. Third is lower coarse which has a gentle slope and is almost flat. The river channel is usually at its widest and deepest here because the amount of water flowing within the river is at its greatest. Oxbow lakes are conspicuous features in the lower coarse of the river apart from deltas where the river drains into seas.

1.3 Meander Path

The course of the river comprises of three distinct path. First is the upper or the mountain course where the river originates. Here, the velocity of the water is faster due to steep gradient and the floor of the valley is narrow and the valleys are V shaped. Second is the middle or the valley coarse where floors are wider and the sides of the valley are more gently sloping. Here, the velocity of the river is slower than the upper stage. Meanders are typical landforms found in this stage of the river. Third is lower coarse which has a gentle slope and is almost flat. The river channel is usually at its widest and deepest here because the amount of water flowing within the river is at its greatest. Oxbow lakes are conspicuous features in the lower coarse of the river apart from deltas where the river drains into seas. Meanders are formed in the middle coarse of the river. The rivers flow through almost a flat plane in the middle coarse as water flowing under gravity seldom flows straight for any long distance, a winding coarse soon develops the irregularities of the ground force the river to swing in great s-shaped curves forming horseshoe like loops called meanders.

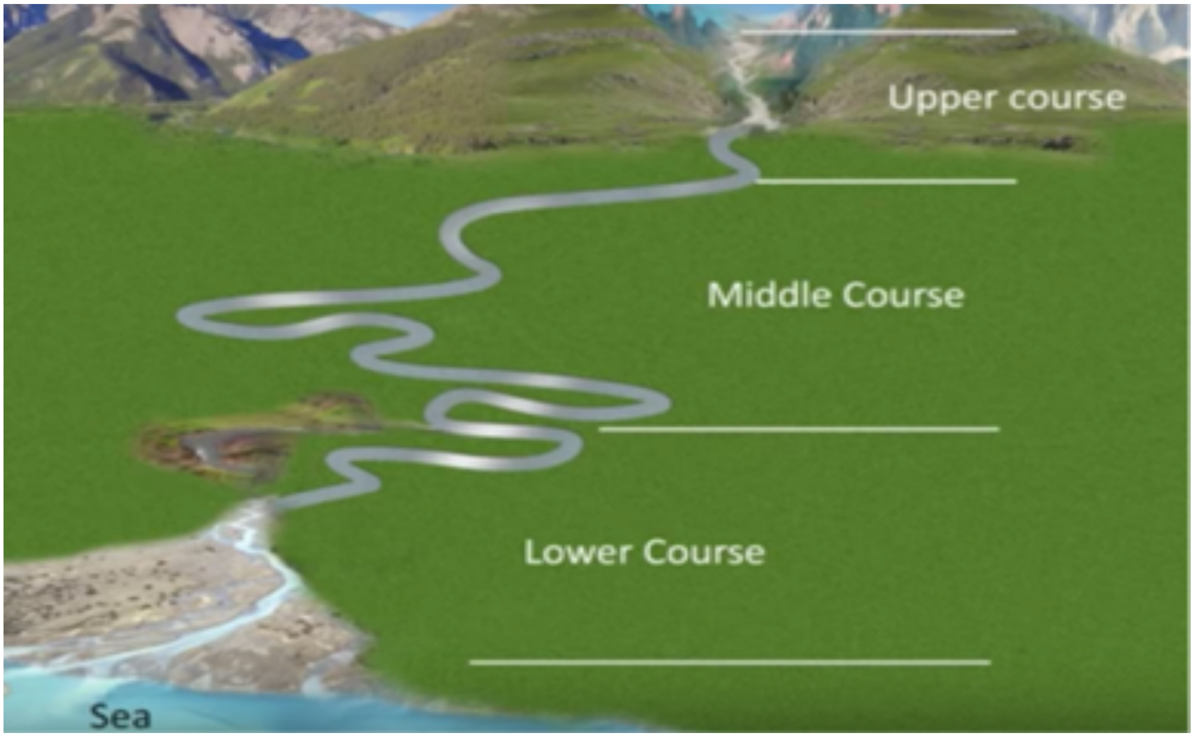

Fig 1.2. Different paths of meandering river

The water in a river channel flows faster round the outside bend of the river due to centrifugal force, whereas it is relatively slow on the inside was bends. The fast flowing water on the outside bend erodes the riverbank whereas slow moving water deposit sand and mud on the inside bend.

The water in a river channel flows faster round the outside bend of the river due to centrifugal force, whereas it is relatively slow on the inside was bends. The fast flowing water on the outside bend erodes the riverbank whereas slow moving water deposit sand and mud on the inside bend. At the site of erosion on the outside bend a steep river cliff is created. The water piles up on the outside bend due to centrifugal force, a bottom current is set up in a cock screw motion which deposits the eroded material on the inner bank forming a slip of slope. In the lower coarse of the river a meander becomes very much pronounced. The outside bend gets eroded rapidly and the meander loop becomes almost a complete circle, gradually the narrow necks of the loop erode further and interest each other and the river cuts through intersection to flow in a straight path as a result the meander loop is abandoned and it forms an oxbow lake. The oxbow lake later turns into a swamp and subsequent floods may silt up the lake, it becomes marshy and eventually dries up.

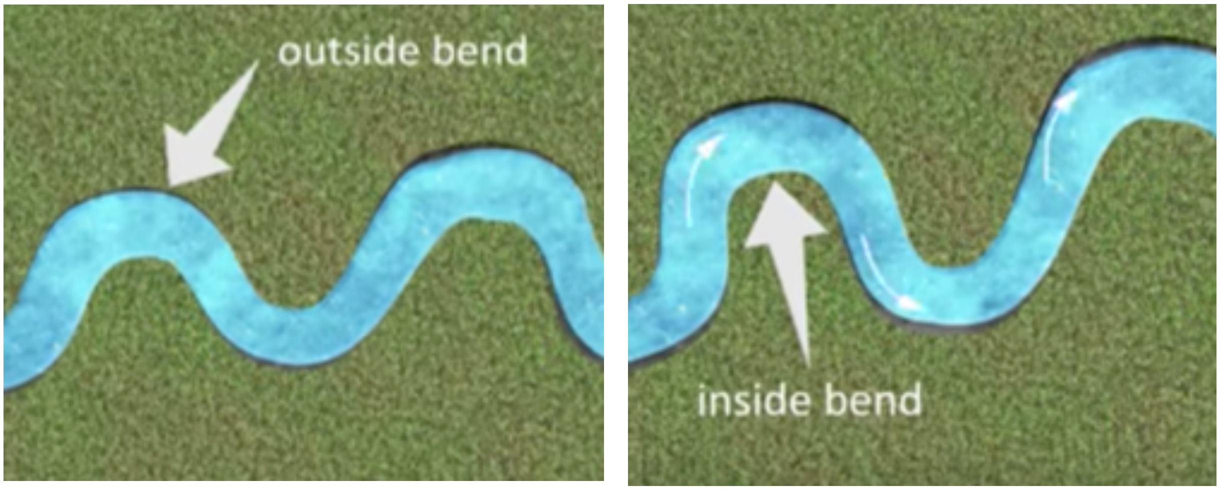

Fig 1.3. Outside and inside bends of the meandering channel

1.4 Boundary Shear Stress

When the water flows in the river or in the open channel it is certainly opposed by river bed and side wall of the channel. The force resisting the flow of water by side wall and bed of the river is called boundary shear force. Boundary shear force is the hydrodynamic force which tangential components act on the walls and bed of the channel. This shear force act on the wetted perimeter of the channel and flow of water is directly dependent on it. Shear stress distribution depends on many factors such as slope of the channel, depth of the channel, shape and cross section of the channel, sediment concentration and wall roughness etc. Sinuosity of the channel is one of the critical parameter for the distribution of shear stress in channel. The water in the sinusoidal river has greater boundary shear stress which is responsible for withering the surface of river channel. Because of large shear stress on bed and walls of the channel and the sediments carried by water, side walls of the channel get eroded and that leads to inception of river meanders. Boundary shear stress is not uniform throughout the channel and and it varies at different angles and also at different depth of the channel. In this research work steps has been taken to calculate the boundary shear stresses at different angles and also to study the variation of shear stress at different discharge. Shear stress distribution depends on many factors such as slope of the channel, depth of the channel, shape and cross section of the channel, sediment concentration and wall roughness etc. Sinuosity of the channel is one of the critical parameter for the distribution of shear stress in channel.

Secondary flow is one of the major factor responsible for the augmentation of the boundary shear stress in flowing river. Shear stress distribution depends on many factors such as slope of the channel, depth of the channel, shape and cross section of the channel, sediment concentration and wall roughness etc. Sinuosity of the channel is one of the critical parameter for the distribution of shear stress in channel. When the secondary flow moves towards the wall then the boundary shear stress increases and when the secondary flow moves away from the wall boundary shear stress decreases.

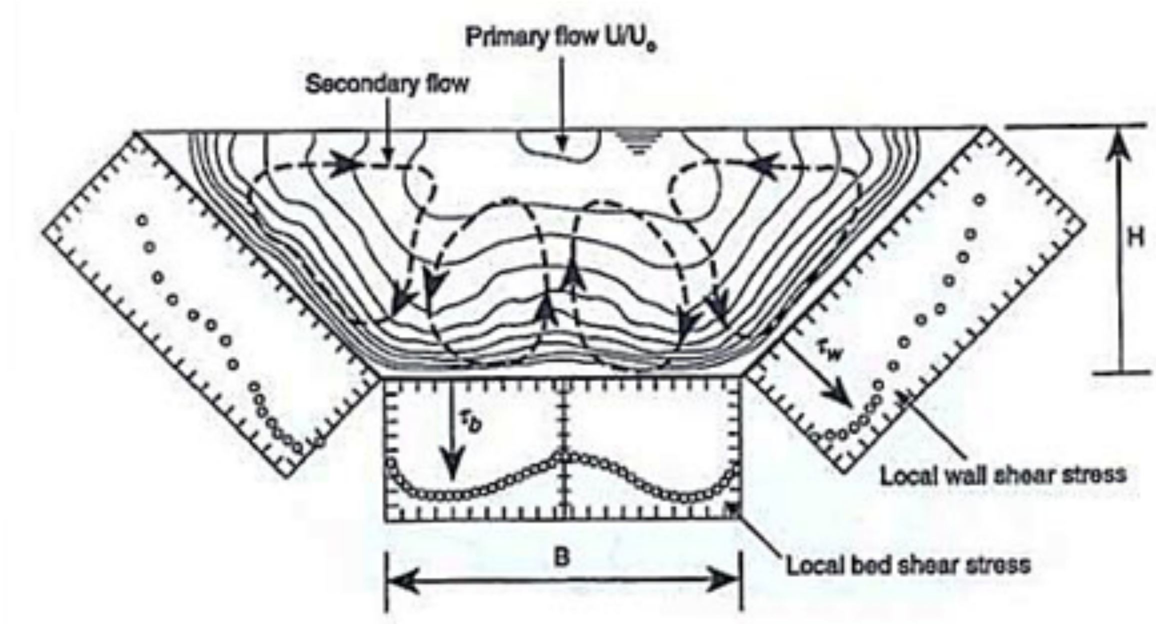

Fig 1.4. Shear stress distribution along walls and bed in trapezoidal channel.

1.5 Computational Fluid Mechanics

Computational fluid dynamics is the study of how fluids flow using computer softwares. The governing mathematical equations that explain how fluids flow are typically complicated to solve by conventional methods. CFD is used in various sectors like aerospace, automobile, renewable energy sectors etc. It is used in the analysis of any object or thing that has the tendency to flow. There are number of CFD software available that have ability to do modelling and analysis of multiphase flow. Some of the softwares are COMSOL, Autodesk, ANSYS etc. In this research work approach has been taken to analyse the behaviour of the meandering channel using CFD tool, known as ANSYS FLUENT. The precision of the computational analysis and its results depends on meshing of the model and numbers of iterations done. In this research CFD model has been made to investigate to impact on the meandering channel and to study the changes in boundary shear stress along the length of the channel or throughout the meander path from one end of the apex to the other end of the apex. The CFD model created for an open-channel was validated by looking at the velocity profile acquired by the numerical simulation with the actual measurement did by experimentation in the same channel utilising Preston tube. The CFD model has been used to investigate the impacts of flow meandering of the channel, and to study the varieties in velocity profiles along the meander path from one bend apex to the other. The reproduced the simulated flow field in each case is compared with corresponding laboratory measurements of velocity distribution. Distinctive models are utilised to unravel Navier-Stokes mathematical equations which are the governing equations for any fluid flow. Finite volume method is applied to discretise the governing equations. The precision of computational results essentially relay upon the mesh quality and the model used to simulate the flow.

The CFD model has been used to investigate the impacts of flow meandering of the channel, and to study the varieties in velocity profiles along the meander path from one bend apex to the other. The reproduced the simulated flow field in each case is compared with corresponding laboratory measurements of velocity distribution. Distinctive models are utilised to unravel Navier-Stokes mathematical equations which are the governing equations for any fluid flow. Finite volume method is applied to discretise the governing equations. The precision of computational results essentially relay upon the mesh quality and the model used to simulate the flow. It is used in the analysis of any object or thing that has the tendency to flow. There are number of CFD software available that have ability to do modelling and analysis of multiphase flow. Some of the softwares are COMSOL, Autodesk, ANSYS etc. In this research work approach has been taken to analyse the behaviour of the meandering channel using CFD tool, known as ANSYS FLUENT. The precision of the computational analysis and its results depends on meshing of the model and numbers of iterations done. In this research CFD model has been made to investigate to impact on the meandering channel and to study the changes in boundary shear stress along the length of the channel or throughout the meander path from one end of the apex to the other end of the apex. The CFD model created for an open-channel was validated by looking at the velocity profile acquired by the numerical simulation with the actual measurement did by experimentation in the same channel utilising Preston tube. The CFD model has been used to investigate the impacts of flow meandering of the channel, and to study the varieties in velocity profiles along the meander path from one bend apex to the other. The reproduced the simulated flow field in each case is compared with corresponding laboratory measurements of velocity distribution. Distinctive models are utilised to unravel Navier-Stokes mathematical equations which are the governing equations for any fluid flow. Computational fluid dynamics is the study of how fluids flow using computer softwares. The CFD model has been used to investigate the impacts of flow meandering of the channel, and to study the varieties in velocity profiles along the meander path from one bend apex to the other. The governing mathematical equations that explain how fluids flow are typically complicated to solve by conventional methods. CFD is used in various sectors like aerospace, automobile, renewable energy sectors etc. It is used in the analysis of any object or thing that has the tendency to flow. Numerical modelling is the computer based technique in which iteration is required to calculate the approx result. More number of iteration will lead to more accurate result. Numerical technique helps to investigate and validate the experiments done in laboratory. It is not necessary to do prior experiment to investigate the impact of the model. The results and its value depends upon various parameters like size of the channel, its length, width and height and also the sinuosity of the channel. Apart from these physical parameters some of the others parameters are also responsible for the variation in results such as flow velocity, Reynolds number, material used etc. for the result to be more accurate maximum number of iteration is important. The results in ansys fluent comes as per the different axis of the model which is been made by meshing in the ansys software. The precision of the results not only depends upon the number of iterations as mentioned above but also on the meshing of the model. Fine meshing done will lead to precise results. In the experiment perform the meshing of the model is fine and the iterations are also done different number of time to see the various results and the most approximate results is been taken to study the variation in the behaviour in the boundary shear stress of the meandering river with change in discharge. As per the conventional technique and experimentation the value of shear stress increases with the increase in discharge or velocity of the fluid. The same has been noticed in the analysis done by using ansys fluent software.

The process of numerical analysis methodology has various steps and sub-steps. First step is the identification of the problem and defining the modelling goals as well as defining the domain of the model. Second step involves the preprocessing of the model in which geometry is setup by creating a solid model to represent the domain and designing or creating the mesh of the model. Third step involves the application of the physics in which first step is to define the flow condition of the model whether it is laminar or turbulent and then specification of the boundary condition is done appropriately as well as specification of the temporal condition is also done appropriately. This step also include discretise the governing equation by using different numerical scheme, iteration of the equation is also done number of times to reach the accuracy, initialisation controlling and monitoring of the solution also involve in this step. Finally the visualisation and examining of the result is done on the x-axis and y-axis from the drawn contour lines.

Computational Fluid Mechanics and its equations are used to calculate or analyse the result of the fluid model. Formation of meander of the river is not a short process but it takes hundreds and thousands of river to change its path and form the meander path. It sometimes becomes difficult to analyse the behaviour of the meandering river or the straight flowing river which turns into a meander river. Experimentation and analysis of the river river and its changes and formation as a meander river is indeed a long process and it is not possible to look at the behaviour of this in laboratory. So, sometimes it becomes important to use the technique of numerical analysis and computational fluid mechanics to experimenting and analysing the long term changes like meandering of river. Finding boundary shear stress distribution on different surfaces of the channel becomes very easy with the help of numerical technique. Although calculating the silt deposits on the sides and bed of the meandering river becomes bit complex as it requires the flow of silt with along with the flow of water or fluid. The scope of the work is not limited as it involves various parameters and their calculations. As mentioned earlier the results depends on the number of iterations done and also on the meshing of the model but it also depends on the governing equation and the method used. In this particular analysis finite volume method is used to calculate the governing equations. Various methods used do not make large impact on the variation of the result but it do responsible for the changes in results by the small amount. So, it is of not major concern. So, while designing and analysing the model, two things are of utmost important that is the meshing of the channel, which should be fine and number of iteration done, which should be maximum.

1.6 Numerical Modelling

Numerical modelling is the computer based technique in which iteration is required to calculate the approx result. More number of iteration will lead to more accurate result. Numerical technique helps to investigate and validate the experiments done in laboratory. It is not necessary to do prior experiment to investigate the impact of the model. The results and its value depends upon various parameters like size of the channel, its length, width and height and also the sinuosity of the channel. Apart from these physical parameters some of the others parameters are also responsible for the variation in results such as flow velocity, Reynolds number, material used etc. for the result to be more accurate maximum number of iteration is important. The results in ansys fluent comes as per the different axis of the model which is been made by meshing in the ansys software. The precision of the results not only depends upon the number of iterations as mentioned above but also on the meshing of the model. Fine meshing done will lead to precise results. In the experiment perform the meshing of the model is fine and the iterations are also done different number of time to see the various results and the most approximate results is been taken to study the variation in the behaviour in the boundary shear stress of the meandering river with change in discharge. As per the conventional technique and experimentation the value of shear stress increases with the increase in discharge or velocity of the fluid. The same has been noticed in the analysis done by using ansys fluent software.

The process of numerical analysis methodology has various steps and sub-steps. First step is the identification of the problem and defining the modelling goals as well as defining the domain of the model. Second step involves the pre-processing of the model in which geometry is setup by creating a solid model to represent the domain and designing or creating the mesh of the model. Third step involves the application of the physics in which first step is to define the flow condition of the model whether it is laminar or turbulent and then specification of the boundary condition is done appropriately as well as specification of the temporal condition is also done appropriately. This step also include discretise the governing equation by using different numerical scheme, iteration of the equation is also done number of times to reach the accuracy, initialisation controlling and monitoring of the solution also involve in this step. Finally the visualisation and examining of the result is done on the x-axis and y-axis from the drawn contour lines.

The process of numerical analysis methodology has various steps and sub-steps. First step is the identification of the problem and defining the modelling goals as well as defining the domain of the model. Second step involves the preprocessing of the model in which geometry is setup by creating a solid model to represent the domain and designing or creating the mesh of the model. Third step involves the application of the physics in which first step is to define the flow condition of the model whether it is laminar or turbulent and then specification of the boundary condition is done appropriately as well as specification of the temporal condition is also done appropriately.

CHAPTER 2
LITERATURE REVIEW

2. LITERATURE REVIEW

2.1 PREVIOUS WORK ON BOUNDARY SHEAR STRESS

In straight channel , the longitudinal velocity in the channel is generally faster. This results a shear layer at the interface of straight channel. Due to the presence of this shear layer , the flow in the straight channel decreases because of the effect of faster flows. This result shows that the flow decreases the whole discharge of the cross section.

Knight and Macdonald (1979) contemplated that the resistance of the channel bed differed by artificial strip roughness components, and estimations made of the divider and bed shear stresses. The distribution of velocity and boundary shear stress in rectangular flume was analysed tentatively, and the impact of fluctuating the bed roughness and aspect ratio were accessed.

Knight (1981) gave an empirically determined equation that presented the percentage of the shear force carried by the walls as a component of the breadth/depth proportion and the proportion between the identical roughness sizes for the bed and the walls. The outcomes were contrasted and other accessible information for the smooth channel case and few differences noted. The systematic reduction in the shear force conveyed by the walls with expanding breadth/depth proportion and bed roughness was shown. Further equations were exhibited giving the mean wall and bed shear stress variety with aspect ratio and roughness parameters.

Knight and Patel (1981) reported a part of laboratory examination results concerning the distribution of boundary shear stress in smooth close conduits of a rectangular cross section for an aspect ratio around 1 and 10. The distributions were shown to be influenced by the number and state of the secondary flow cells, which, therefore, depended essentially upon the aspect ratio. For a square cross section with 8 symmetrically arranged secondary flow cells, a two fold top in the distribution of the boundary shear stress along every wall was shown to dislodge the maximum shear stress far from the centre position towards every corner. For square cross portions, For a square cross segment with 8 symmetrically arranged optional stream cells, a twofold top in the appropriation of the limit shear push along every divider was demonstrated to uproot the most extreme shear stretch far from the inside position towards every corner. For rectangular cross sections, the quantity of secondary flow cells increased from 8 by augmentations of 4 as the aspect ratio increased, bringing on alternate perturbations in the boundary shear stress distribution at positions where there were adjacent contra-rotating flow cells. Equations were presented for the most extreme, centreline and mean boundary shear stress on the duct walls in terms of aspect ratio.

Leighly (1982) studied the boundary shear stress distribution in open channel flow by using conformal mapping. He pointed out that, in the absence of secondary currents, the boundary shear stress acting on the bed must be balanced by the downstream component of the weight of water contained within the bounding orthogonal shear layer

Cruff (1985) estimate the boundary shear stress by the use of the Preston tube technique as well as the Karman Prandtl logarithmic velocity law from uniform flow in rectangular channel. Though he did not calculate boundary shear stresses distribution in rectangular channel with over bank flow, mainly his work was known for a method which was help to calculate the apparent shear stress and momentum transfer between a channel and its flood plain.

Ghosh and Jena (1982) gives the boundary shear distribution for rough and smooth in compound channel. Using the Preston tube technique combined with the Patel calibration, they found the boundary shear distribution along the wetted perimeter of the total channel for various depths of flow. From the analysis, it is clearly shows that the maximum shear stress on the channel bed and approximately midway between the centre line and corner.

From the experimental analysis the shear distribution is probable to estimate τ_C' the average shear stress in the channel. It is analysed that roughening the total periphery of the boundary shear in a meandering channel could be redistributed with the maximum shear at the channel bed.

Rajaratnam and Ahmadi (1989) were doing experimental work in a channel 18.29 meters long, 1.22 meters wide and 0.9 meters deep. Here, they recorded the velocity traverses and boundary shear stresses. They analysed velocity profiles at different depths in the channel exhibited similarity.

Bathurst *et al.* (1989) obtainable the field data for calculating the bed shear stress in a curved river and it is obtained that the bed shear stress distribution is affected by both the location of core of the main velocity and the structure of secondary flows.

Knight and Demetriou (1993) carried out series of experiments in straight symmetrical compound channels to find the characteristics of discharge, boundary shear stress and boundary shear force distributions. Equations to calculate the percentage of shear force carried by the flood plain were being proposed. The apparent shear force was observed to be highest at the lower flow depth. For high floodplain widths for vertical interface between main channel and flood plain the apparent shear force was also found to be higher.

Knight and Patel (1995) take the laboratory experiments results which concerning the boundary shears distribution in smooth rectangular cross section for different aspect ratios between 1 and 10. The boundary shear distributions were shown to be subjective by the number and shape of the secondary flow cells, which, was depended mainly on the aspect ratio. Equations were given for the maximum, centreline and mean boundary shear stresses on the channel walls in terms of the aspect ratio.

Knight, Yuan and Fares (1995) gives the experimental data of SERC-FCF about the boundary shear stress distributions in meandering channels through the path of one complete wave length. They also reported the experimental data on surface topography, velocity vectors, and turbulence for meandering channel. They studied the effects of channel sinuosity, secondary currents, and cross section geometry on the value of boundary shear in meandering channels and presented a momentum-force balance for the flow.

Shiono, Muto, Knight and Hyde (1999) presented the secondary flow and turbulence data using two components Laser- Doppler anemometer. They developed the turbulence models, and studied the behaviour of secondary flow for both in bank and over bank flow conditions. They divided the channel into three sub areas, namely; (i)the main channel below the horizontal interface (ii)The meander belt above the interfaces and (iii)The area outside the meander belt of the flood plain. They investigated the energy losses for compound meandering channels resulting from boundary friction, secondary flow, turbulence, expansion and contraction. They reported that the energy loss at the horizontal interface due to shear layer, the energy loss due to bed friction and energy loss due to secondary flow in lower main channel have the significant contribution to the shallow over-bank flow. They also concluded that the energy loss due to expansion and contraction in meander belt have the significant contribution to the high over-bank flow.

Knight and Sterling (2000) analysed the boundary shear distribution in a circular channels flowing partially full with and without a smooth flat bed using Preston-tube technique. The results have been analysed that the variation of local shear stress with perimeter distance and the percentage of total shear force acting on wall or bed of the channel. The %SFW results have been validating with Knight‟s (1981) empirical formula for prismatic channels. **Patra and Kar (2000)** reported the test results concerning the boundary shear stress, shear force, and discharge characteristics of compound meandering river sections composed of a rectangular main channel and one or two flood plains disposed1of to its sides. They used five dimensionless channel parameters to form equations representing the total shear force percentage carried by flood plains. A set of smooth and rough sections were studied with aspect ratio varying from 2 to 5. Apparent shear forces on the assumed vertical, diagonal, and horizontal interface plains were found to be different from zero at low depths of flow and changed sign with increase in depth over floodplain. They proposed a variable-inclined interface for which apparent shear force was calculated as zero. They presented empirical equations predicting proportion of discharge carried by the main channel and floodplain.

Knight and Sterling (2000) studied the appropriation of boundary shear stress in circular conduits flowing mostly full with and without smooth level bed for an information extending a from 0.375<F<1.96 and 6.5*104<R<3.42*105, utilising Preston-tube technique. The distribution of boundary shear stress is demonstrated to rely on upon geometry & Froude no. The outcomes have been examined as far as variety of local shear stress with edge separation and the rate of aggregate shear force following upon wall or bed of the course. The % SFW results have been indicated to concur well with Knight's (1981) exact equation for prismatic channels. The inter dependency of secondary flow and boundary shear stress has been made and its implications for sediment transport have been analysed.

Jin *et. al.* (2004) proposed semi analytical model for prediction of boundary shear distribution in straight open channels. Secondary Reynolds stress terms were involved to develop the simplified stream-wise vorticity equation. An empirical model was generated for computing the effect of the channel boundary on shear stresses. In the calculation of the boundary shear distribution in trapezoidal open channels, the model predictions were in good agreement with the experimental data.

Duan (2004) found that a 2D model could be better because of being computationally cost-effective for parametric analyses needed by policy and management planning as well as preliminary design applications. Because they compared the flow analysis in mildly and sharply curved or meandering channels through the use of depth averaged 2-D model and full 3-D model and established that the last one is more skilled than the previous in taking the flow fields in meandering channels. Finally the author concluded that the 1D, 2D and 3D numerical models should be integrated and cost effectiveness.

Patra and Kar (2004) reported the test results concerning the flow and velocity distribution in meandering compound river sections. Using power law they presented equations concerning the three-dimensional variation of longitudinal, transverse, and vertical velocity in the main channel and flood plain of meandering compound sections in terms of channel parameters. The results of formulations compared well with their respective experimental channel data obtained from a series of symmetrical and unsymmetrical test channels with smooth and rough surfaces. They also verified the formulations against the natural river and other meandering compound channel data.

Khatua (2008) extended the work of **Patra and Kar (2000)** to meandering compound channels. Using five parameters (sinuosity S_r, amplitude, relative depth, width ratio and aspect ratio) general equations representing the total shear force percentage carried by flood plain was presented. The proposed equations are simple, quite reliable and gave good results with the observed data for straight compound channel of **Knight and Demetriou (1983)**as well as for the meandering compound channel.

Khatua (2010) reported the distribution of boundary shear force for highly meandering channel having distinctly different sinuosity and geometry. Based on the experimental results, the interrelationship between the boundary shear, sinuosity and geometry parameters has been shown. The models are also validated using the well published data of other investigators.

2.4 PREVIOUS RESEARCH ON NUMERICAL MODELLING

The features characterised in open channel flow result from the complex interaction between the fluid and a number of mechanism including shear stress along the channel bed and walls, friction, gravity and turbulence. As numerical hydraulic models can significantly reduce costs associated with the experimental models, therefore in recent decades the use of numerical modelling has been rapidly expanded. With widely spread in computer application, interest has risen in applying more techniques providing more accurate results. In other fluid flow fields such as aeronautics and thermodynamics the implementation of more complex models has represented the advances in computer technology and 3D models are now commonly used.aHowever in open channel flow this conversion has not occurred as rapidly than other sector of engineering and most hydraulics models are either 1D or 2D with very few application of 3D models.aIn this work the application of Computational Fluid Dynamics (CFD) package to open channel flow has been considered. The software includes various models to solve general fluid flow problems. Across the globe various numerical models such standard k-ε model, non-linear k-ε model, k-ω model, algebraic Reynolds stress model (ASM), Reynolds stress model (RSM) and large eddy simulation (LES) have been implemented to simulate the complex secondary structure in open channel flow. The standard k- ε model is an isotropic turbulence closure but fails to reproduce the secondary flows. Although nonlinear k- ε model can simulate secondary currents successfully in a compound channel, it cannot accurately capture some of the turbulence structures. Reynolds stress model (RSM) is very effective in computing the time-averaged quantities and requires much less computing cost. RSM computes Reynolds stresses by directly solving Reynolds stress transport equation but its application to open channel is still limited due to the complexity of the model. Large eddy simulation (LES) solves spatially-averaged Navier-Stokes equation.aLarge eddies are directly resolved, but eddies smaller than mesh are modelled.aThough LES is computationally expensive to be used for industrial application but can efficiently model nearly all eddy sizes. The work of previous researchers regarding the advancements in numerical modelling of open channel flow has been listed below.

Giuseppe and Pezzinga (1994) used the k-ε model to analyse the problem of prediction of uniform turbulent flow in compound channel. This model is useful to predict the secondary currents, caused anisotropy of normal turbulent stresses which are important features of the flow in compound channel as we can determine transverse momentum transfer. He made comparison which shows that the model predicts with accuracy the distribution of primary of the velocity component, the secondary circulation and the discharge distribution.

Cokljat and Younis, Basara and Cokljat (1995) gave the RSM for numerical simulation of free surface flows in a rectangular main channel and a compound channel. They found good agreement between predicted and measured data.

Thomas and Williams (1995) gave description of a LES of steady uniform flow in asymmetric compound channel of trapezoidal cross section with flood plains at Reynolds's number of 430000.This simulation helps to predict the bed stress distribution, velocity distribution and secondary circulation across the flood plain by interacting with main channel and flood plain.

Salvetti et.al(1997) has done LES simulation at relatively large Reynolds number which in turn gives the result of bed shear, secondary motion and vortices well comparable to experimental results.

Ahmed Kassem, Jasim Imran and Jamil A. Khan (2003) examined from the three dimensional modelling of negatively buoyant flow in a diverging channel with a slanting bottom.aThey modified the k- turbulence model for the lightness impact and Bouusinesq close estimation for the Reynolds- averaged equations in diverging channels.

Lu et al. (2004) connected a three-dimensional numerical model to reenact secondary flows the distribution of bed shear stress, the longitudinal and transversal changes of water depth and the distribution of velocity components at 180° bend utilising the standard k- turbulence model.

Sugiyama H,Hitomi D.saito T.(2006) developed turbulence model which includes transport equation of turbulent energy and dissipation along with an algebraic stress model based on the Reynolds stress transport equation. They have demonstrated that the fluctuating vertical Velocity approaches zero close to the free surface. Furthermore, the compound meandering open channel was elucidated to some degree based on computed results. As an after effect of the investigation, the present algebraic Reynolds stress model is shown to be able to reasonably predict the turbulent flow in compound meandering open channel.

Bodnar and Prihoda (2006) exhibited a numerical recreation of the turbulent free surface flow by utilising the k- turbulence model and analysed the way of non-linearity of water surface slant at a sharp bend.

Booij (2003) and VanBalen et al. (2008) displayed the flow design at a mildly bended 180 degree twist and evaluated the secondary flow structure utilising Large Eddy Simulation (LES) model.

Cater and Williams (2008) reported an unequivocal Large Eddy Simulation of turbulent flow In a long compound open channel with one floodplain. The Reynolds number is pretty about 42,000 and the free surface was managed as totally deformable. The results are in simultaneousness with test estimations and support the use of high spatial determination and a vast box length interestingly with a past re enactment of the same geometry. A discretionary flow is perceived at the internal corner that endures and grows the bed weight on the floodplain.

Jing, Guo and Zhang (2009) reenacted a three-dimensional (3D) Reynolds stress model (RSM) for compound meandering channel flows. The velocity fields, wall shear stresses, and Reynolds stress are ascertained for a range of input conditions. Great assertion between the simulated results and measurements shows that RSM can effectively anticipate the confounded flow phenomenon.

B. K. Gandhi, H.K. Verma and Boby Abraham (2010) determined the velocity profiles In both the directions under distinctive real flow conditions, as ideal flow conditions seldom exists in the field. 'Fluent', a commercial computational fluid dynamics (CFD) code, has been utilised to numerically model different situations. They examined the impacts of bed slope, upstream bend.

Balen et.al. (2010) performed LES for a bended open-channel flow over topography. It was observed that, despite the coarse technique for representing the ridge shapes, the Qualitative assertion of the test results and the LES results is fairly great. Also, it is observed that in the bend the structure of the Reynolds stress tensor demonstrates an inclination toward isotropy which upgrades the execution of isotropic eddy viscosity closure models of turbulence.

Esteve et.al. (2010) reenacted the turbulent flow structures in a compound meandering Channel by Large Eddy Simulations (LES) utilising the experimental arrangement of Moto and Shiuno (1998). The Large Eddy Simulation is performed with the in-house code LESOCC2. The anticipated flow wise velocities and secondary current vectors and in addition turbulent intensity are in great concurrence with the LDA measurements.

Ansari et.al. (2011) decided the distribution of the bend and side wall shear stresses in trapezoidal channels and examined the effect of the variety of the slant angles of the side walls, aspect ratio and composite roughness on the shear stress distribution. The outcomes demonstrate a noteworthy contribution on secondary currents and overall shear stress at the boundaries

Rasool Ghobadian and Kamran Mohammodiun (2011) recreated the subcritical flow Pattern in 180° uniform and convergent open-channel curves utilising SSIIM 3-D model with Maximum bed shear stress. They noticed toward the end of the convergent bend, bed shear stress show higher values than those in the same locale in the channel with a uniform twist.

Khazaee & M. Mohammadiun (2012) explored three-dimensional and two phase CFD model for flow distribution in an open channel. He completed the Finite volume method (FVM) with a dynamic Sub grid scale for seven instances of distinctive aspect ratios, different inclination angles or slopes and converging diverging condition.

Omid Seyedashraf, Ali Akbar Akhtari & Milad Khatib Shahidi (2012) reached in a conclusion that the standard k-ε model has the ability of catching particular flow features in open channel curves more precisely. Looking at the location of the minimum velocity occurrences in a customary sharp open channel bend, the minimum velocity occurs close to the inward bank and inside the separation zone along the meandering.

Larocque, Imran, Chaudhry (2013) displayed 3D numerical recreation of a dam-break Flow utilising LES and k- ε turbulence model with following of free surface by volume-of-fluid model. Results are compared with published experimental data on dam break flow through incomplete break and additionally with results acquired by others utilising a shallow water model. The outcomes demonstrate that both the LES and the k –ε displaying palatably imitate the fleeting variety of the measured bottom pressure. Nonetheless, the LES model catches better the free surface and velocity variety with time.

Ramamurthy et al. (2013) reenacted three-dimensional flow design in a sharp bend by utilising two numerical codes alongside different turbulent models, and by comparing the numerical results with results approved the models, and guaranteed that RSM turbulence model has a better agreement with experimental results.

Mohanta (2014)agave the Flow Modelling of a Non Prismatic compound channel By Using CFD. He used the large eddy simulation model to accurately predict the flow features, specifically the distribution of secondary circulations both for in-bank channels as well as over bank channels at varying depth and width ratios in symmetrically converging flood plain compound sections .

Knight (2014) gave an empirically determined equation that presented the percentage of the shear force carried by the walls as a component of the breadth/depth proportion and the proportion between the identical roughness sizes for the bed and the walls. The outcomes were contrasted and other accessible information for the smooth channel case and few differences noted. The systematic reduction in the shear force conveyed by the walls with expanding breadth/depth proportion and bed roughness was shown. Further equations were exhibited giving the mean wall and bed shear stress variety with aspect ratio and roughness parameters.

B. K. Gandhi, H.K. Verma and Boby Abraham (2015) determined the velocity profiles .In both the directions under distinctive real flow conditions, as ideal flow conditions seldom exists in the field. 'Fluent', a commercial computational fluid dynamics (CFD) code, has been utilised to numerically model different situations. They examined the impacts of bed slope, upstream bend.

Balen et.al. (2015) performed LES for a bended open-channel flow over topography. It was observed that, despite the coarse technique for representing the ridge shapes, the Qualitative assertion of the test results and the LES results is fairly great. Also, it is observed that in the bend the structure of the Reynolds stress tensor demonstrates an inclination toward isotropy which upgrades the execution of isotropic eddy viscosity closure models of turbulence.

Esteve et.al. (2015) reenacted the turbulent flow structures in a compound meandering Channel by Large Eddy Simulations (LES) utilising the experimental arrangement of Moto and Shiuno (1998). The Large Eddy Simulation is performed with the in-house code LESOCC2. The anticipated flow wise velocities and secondary current vectors and in addition turbulent intensity are in great concurrence with the LDA measurements.

Ansari et.al. (2015) decided the distribution of the bend and side wall shear stresses in trapezoidal channels and examined the effect of the variety of the slant angles of the side walls, aspect ratio and composite roughness on the shear stress distribution.

CHAPTER 3
METHODOLOGY

3. METHODOLOGY

3.1 Description of Numerical Model Parameters

In this study, Fluent, a computational Fluid Dynamics simulation tool is used for model verification which is a tree-dimensional form of Navier-Stokes equations. Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses various numerical models and algorithm to analyze and solve issues that include fluid flow. Computer systems are utilized to perform the calculations needed to simulate the interaction of fluid flows and gases with the surfaces characterized by boundary conditions. With fast supercomputers, better arrangements can be accomplished the precision and rate of complex simulation situations, for example, transonic or turbulent flows. It has begun around 1960 and with the procedure of change in computer processor speed, CFD simulation is currently demonstrating astonishing exactness. The CFD based on-going examination yields programing that enhances simulation depends combined numerical exactness, demonstrating accuracy and computer expense. When the residuals of the discretized transport equation reach a value of 0.001 or when the solution do not change with further iterations, the numerical solution is converged. For the simulation with a unsteady solver, the distinction in the mass flow rates at the velocity inlet and pressure outlet is observed to be under 0.01% during final solution. Furthermost, various additional time steps are added to check the steadiness of the flow in the last arrangement.

3.2 Turbulence Modelling

"Turbulence is an irregular movement which is shown in liquids and gases when they flow past solid surfaces or even when neighbouring streams of the same fluid flow past over one another." GI Taylor and von Karman, 1937 "Turbulent smooth movement is an unpredictable state of flow in which the various quantities Show an arbitrary variation with time and space coordinates, so that statically distinct average values can be perceived." Hinzee, 1959. The stream in a meandering channel is turbulent in nature. Channel Shape or geometry and gravity force is mostly in charge of the turbulent stream. Turbulent flow is a flow regime described by turbulent and stochastic property changes. This incorporates low momentum dissemination, high energy convection, and rapid variation of force and velocity in space and time. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterised by a high Reynolds Number.

Generally turbulence is a random three dimensional time dependent eddying motion with many large scales eddies. The three dimensional nature of the turbulent flows are decomposed into two different parts i.e. mean part and fluctuation part, which is well known as Reynolds decomposition. The spatial character of the turbulence reveal the eddies with wide range scales. In turbulence, separated fluid structure involved in it creates uncertainty in the prediction of flow variable particularly in meandering channels, turbulent structure are generalized by large shear layer region creates vortices both longitudinal as well as vertical direction. The anisotropy and inhomogeneity of turbulent structure causes secondary current, which creates the vertical dip affects the flow variables. Hence in this study an effort is made to recognize the effect of the turbulence in a meandering channel of an average velocity component and a fluctuating Velocity component given as Instantaneous velocity = mean velocity + fluctuating is given by:

$$u = \bar{u} + u' \tag{1}$$

The Navier-Stokes momentum equation is taken as:

$$\frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \frac{\partial u_i}{\partial x_j}) \tag{2}$$

By substituting $\bar{u} + u'$for u in equation (3.2.2) and averaging the term we get:

$$\overline{\frac{\partial u}{\partial x}} = \overline{\frac{\partial(\bar{u}+u\prime)}{\partial x}} = \frac{\partial \bar{u}}{\partial x} \tag{3}$$

For non-linear function the equation (3.2.1) becomes

$$\frac{\partial(uu)}{\partial x} = \frac{\partial(\overline{uu})}{\partial x} = \frac{\partial \overline{u\prime u}}{\partial x} \tag{4}$$

Now the Navier-Stokes equations become:

$$\frac{\partial \overline{u_i}}{\partial x_j} = 0 \tag{5}$$

3.3 Turbulence Models

1. Algebraic (zero-equation) model.
2. K-ε, RNG k-ε model.
3. Shear stress transport model.
4. K-ω model
5. Reynolds stress transport model (second moment closure).
6. K-ω Reynolds stress.
7. Detached eddy simulation (DES) turbulence model.
8. SST scale adaptive simulation (SAS) turbulence model.
9. Smagorinsky large eddy simulation model (LES).
10. Scalable wall functions.
11. Automatic near-wall treatment including integration to the wall.
12. User-defined turbulent wall functions and heat transfer.

$$\frac{\partial \rho \overline{u_i u_j}}{\partial x_j} = -\frac{\partial y}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j}\right) - \frac{\partial \rho (u'_i u'_j)}{\partial x_j} \qquad (6)$$

The term $\frac{\partial \rho (u'_i u'_j)}{\partial x_j}$ is known as the "Reynolds stress". Due to closure problem of both the equations (5) and (6) we have to come up with ways of replacing the extra terms with other terms that were known or devising ways of calculating these terms. A first attempt at closing the equations is:

$$\frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j}\right) = \frac{\partial \rho (u'_i u'_j)}{\partial x_j} \qquad (7)$$

In equation (7) both terms represent a diffusion of energy. The term $\frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j}\right)$ represents diffusion of energy through viscosity and the other term $\frac{\partial \rho (u'_i u'_j)}{\partial x_j}$ represents the diffusion through turbulence. By defining μ_t as turbulent viscosity, equation (6) becomes:

$$\frac{\partial \rho \overline{u_\iota u_J}}{\partial x_j} = -\frac{\partial y}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\mu + \mu_t)\frac{\partial \overline{u_\iota}}{\partial x_j}\right) \tag{8}$$

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods which make use of turbulence models. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and direct numerical simulation. Most turbulence models are statistical turbulence model, as mentioned below.

$$\frac{\partial(\rho)}{\partial t} = -\frac{\partial(\rho u_i)}{\partial x_i} = S_m \tag{9}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial(p)}{\partial x_i} + \frac{\partial}{\partial x_j}\mu\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] + \frac{\partial(-\rho\overline{u'_\iota u'_J})}{\partial x_j} \tag{10}$$

Where t=time, u_i=ith component of the Reynolds-averaged velocity, x_i=ith axis, ρ =water density, p= Reynolds averaged pressure, g=acceleration due to gravity, μ =viscosity (here it is equal to zero), S_m=mass exchange between two phase (water and air).Here for unsteady solver the time-averaged values of velocities and other solution variables are taken instead of instantaneous values. The term $(-\rho\overline{u'_\iota u'_J})$ is called as Reynolds Stress. To link the mean rate of deformation with Reynolds stresses, Bouusinesq hypothesis is used:

$$-\rho u'_i u'_j = \mu_t\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] \tag{11}$$

Where μ_t=the turbulent viscosity

3.4Numerical Methodology

The process of the numerical simulation of fluid flow using the above equation generally involves four different steps

1. Problem Identification

   (1) Defining Modelling goals

   (2) Identifying the domain to model

2. Pre-Processing

   (1) Creating a solid model to represent the domain (Geometry Setup)

   (2) Design and create the mesh (grid)

3. Solver

   (1) Set up the physics

      a. Defining the condition of flow (e.g. turbulent, laminar etc.)
      b. Specification of appropriate boundary condition and temporal condition.

   (2) Using different numerical schemes to discretize the governing equations.

   (3) Controlling the convergence by iterating the equation till accuracy is achieved

   (4) Compute Solution by Solver Setting.

      a. Initialization

      b. Solution Control

      c. Monitoring Solution

4. Post processing ： (1) Visualizing and examining the results

   (2) X-Y Plots

   (3) Contour Draw

The process of numerical analysis methodology has various steps and sub-steps. First step is the identification of the problem and defining the modelling goals as well as defining the domain of the model. Second step involves the pre-processing of the model in which geometry is setup by creating a solid model to represent the domain and designing or creating the mesh of the model. Third step involves the application of the physics in which first step is to define the flow condition of the model whether it is laminar or turbulent and then specification of the boundary condition is done appropriately as well as specification of the temporal condition is also done appropriately.

This step also include discretise the governing equation by using different numerical scheme, iteration of the equation is also done number of times to reach the accuracy, initialisation controlling and monitoring of the solution also involve in this step. Finally, the visualisation and examining of the result is done on the x-axis and y-axis from the drawn contour lines.

3.5 Creation of Geometry

Table 3.1. Details of Geometrical Parameters of the Channel

S. No	Parameters	Description
1	Channel type	Meandering
2	Dimension of flume	4.0 m x 0.5 m x 15 m
3	Geometry of main channel	Trapezoidal with side slopes 1:1
4	Bed Surface Type	Smooth Bed
5	Cross Section of Channel	0.33 m at Bottom and 0.46 m at Top
6	Bank Full depth	0.065 m
7	Bed Slope of the Main Channel	0.00040146
8	Sinuosity of the Main Channel	4.11
9	Amplitude of the Meandering Channel	1.555 m
10	Wave length of the Meandering Channel	2.162 m

Creation of geometry is based on parameters mentioned in the table. The various parameters used are based on the laboratory models of meandering channel. The meandering channel is a simple meandering channel and the type of bed surface is rigid and smooth bed and the channel is of trapezoidal shape with the slope of 1:1. Apart from these various parameters are given below. The design based on the various parameters of the laboratory based model of meandering channel and it is designed by using ansys fluent. The channel has the free surface symmetry as shown in the figure and channel has two walls on which the shear stress is calculated. Similarly, channel bottom corresponds to bed of the channel and the shear stress is calculated on it at different quantity if discharg.

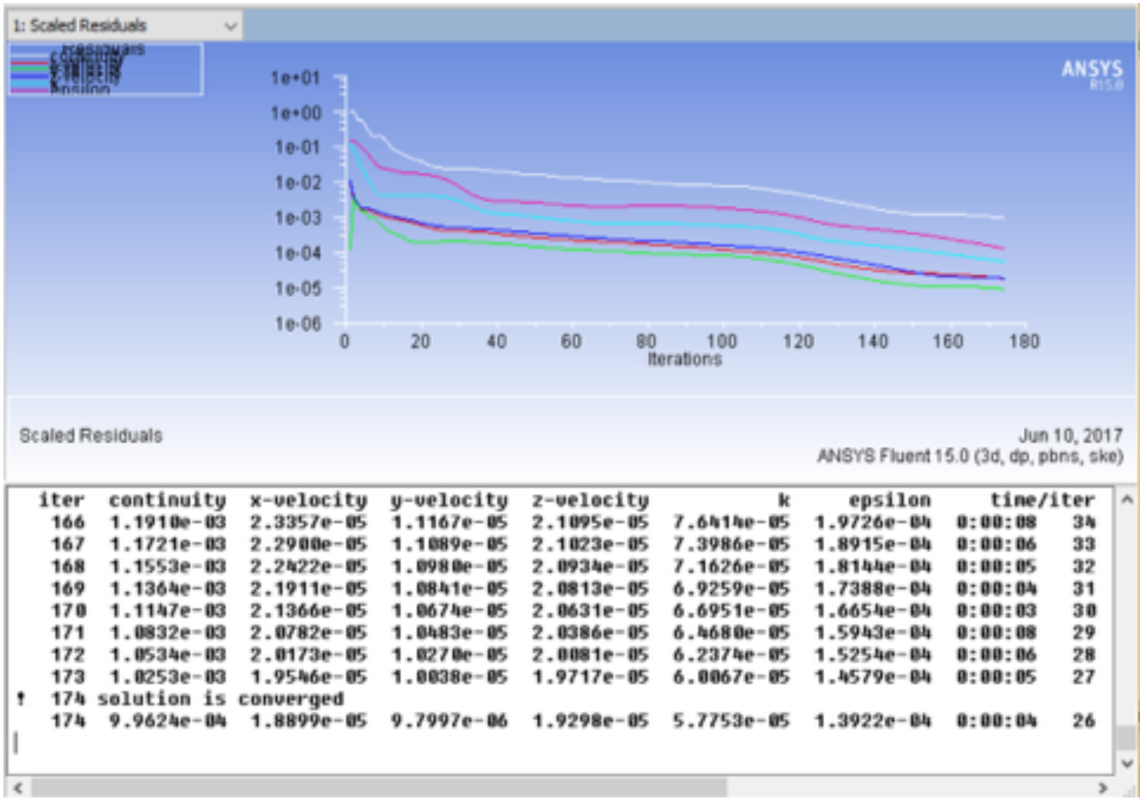

Fig 3.1 Iteration done in Ansys Design Modular

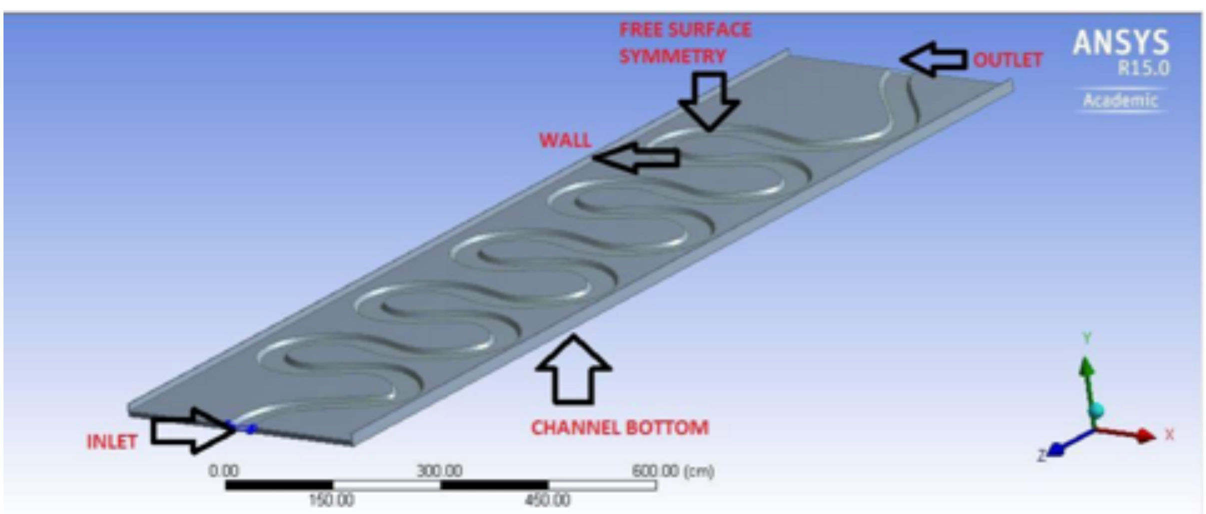

Fig 3.2 Close Geometry of Ansys Design Modular

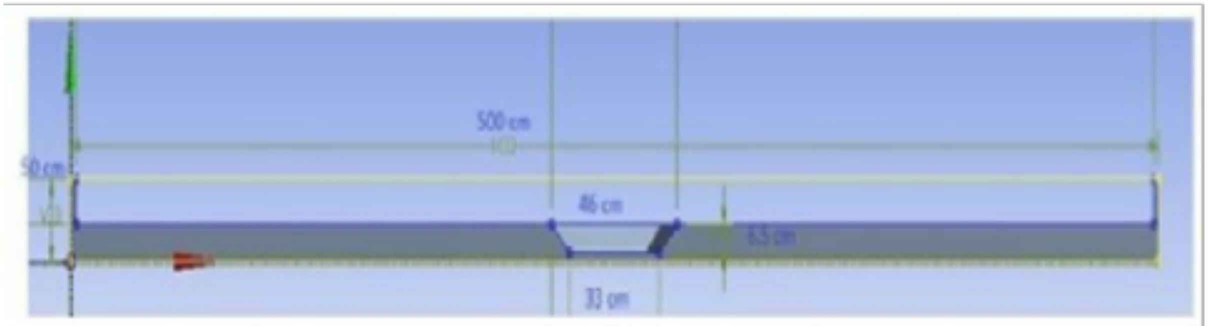

Fig 3.3 Channel cross section in Ansys Design Modular

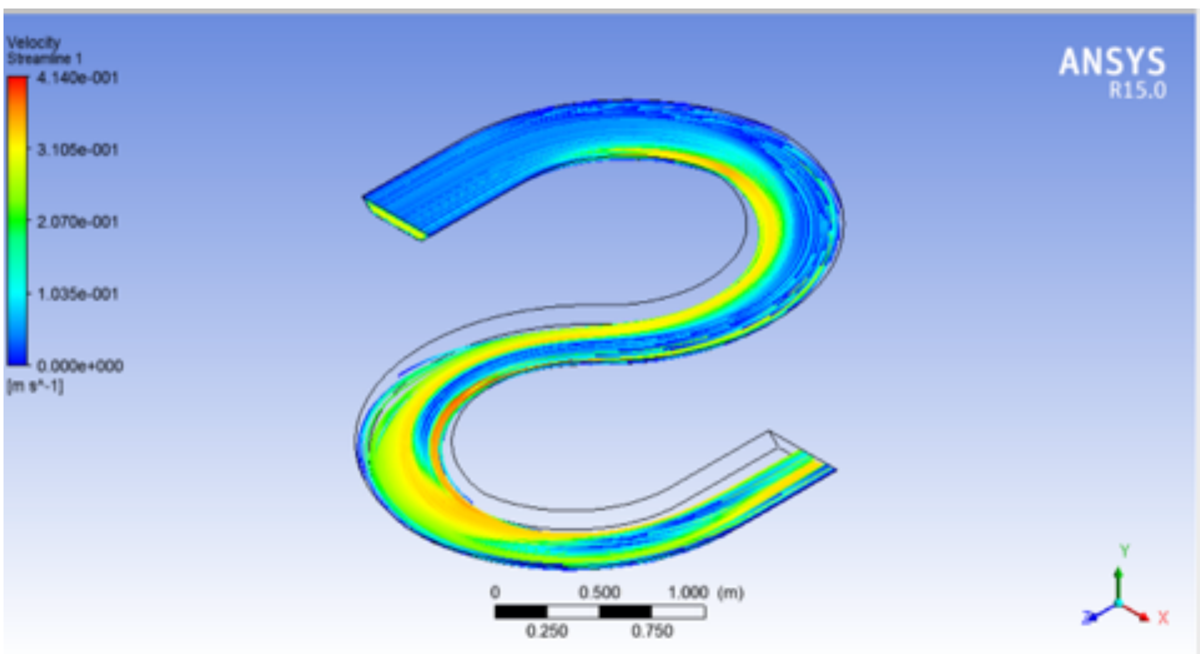

Fig 3.4 Close Geometry of Ansys Design Modular

3.6 Mesh Generation

Second and very most important step in numerical analysis is setting up the grid associated with the construction of geometry. The Navier-Stokes Equations are non-linear partial differential equations, which consider the whole fluid domain as a continuum. In order to simplify the problem the equations are simplified as simple flows have been directly solved at very low Reynolds numbers. The simplification can be made using what is called discretization. Construction of mesh involves discretizing or subdividing the geometry into the cells or elements at which the variables will be computed numerically. By using the Cartesian co-ordinate system, the fluid flow governing equations i.e. momentum equation, continuity equation are solved based on the discretization of domain. The CFD analysis needs a spatial discretization scheme and time marching scheme. Meshing divides the continuum into finite number of nodes. Generally the domains are discretized by three different ways i.e. Finite element, Finite Volume and Finite Difference Method. Finite element method is based on dividing the domain into elements. In finite element method the numerical solutions are obtained by integrating the shape function and weighted factor in an appropriate domain. This method is suitable for both structured and unstructured mesh. But the Finite Volume method divides the domain into finite number of volumes. Finite volume method solves the discretization equation in the centre of the cell and calculates some specified variables. The values of quantities, such as pressure, density and velocity that are present in the equations to be solved are stored at the centre of each volume. The flux into a region is calculated as the sum of the fluxes at the boundaries of that region. As the values of quantities are stored at nodes but not at boundaries this method requires some interpolation at nodes. Generally finite Volume method is suitable for unstructured domain. Whereas finite Difference method is based on approximation of Taylor's series. This method is more suitable for regular domain.

In the whole process of making or creating the mesh, small cells been made. More the number of cells more fine the mesh will be. Fineness of the mesh is also responsible for the accuracy of the result in numerical analysis. The flow in the geometry is made of simple governing equation with low Reynolds number that will lead less impact and will effect the geometry of the model minimum. Momentum and continuity equation have been solved using discretisation. Generally, it is done by three ways that is finite volume, finite element and finite difference method. The various technique mentioned are discussed above also. Figure 3.3 shows the cross section of the channel. For transient problems an appropriate time step needs to be specified. To capture the required features of fluid flow with in a domain, the time step should be sufficiently small but not too much small which may cause waste of computational power and time. Spatial and time discretisation is linked, as evident in the Courant number.

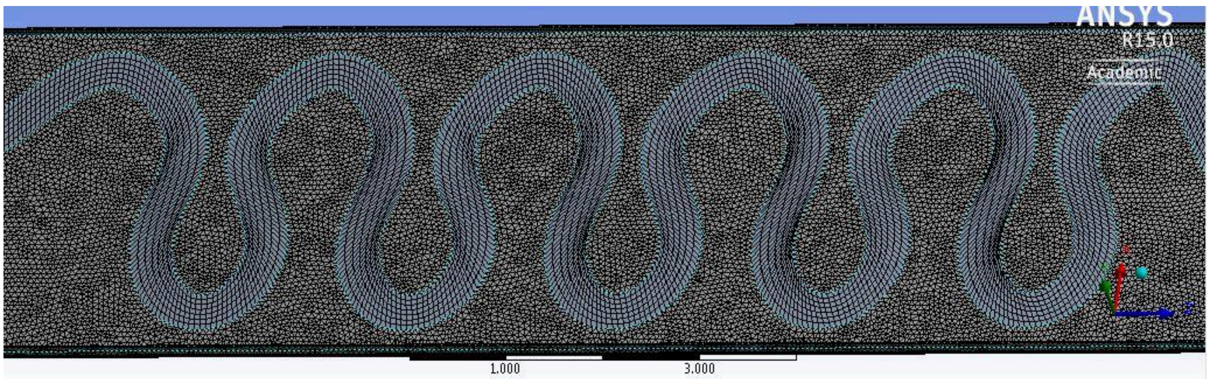

Fig 3.5: Meshing Of The Channel top view

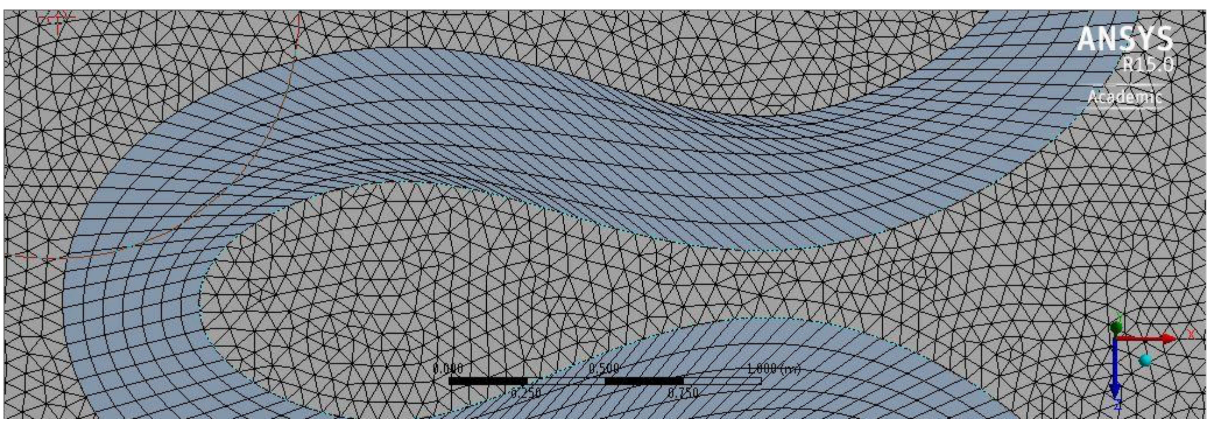

Fig 3.6: Meshing View By Zooming

3.7 Solving for Turbulence

For past three decades various numerical turbulence models such as K-ε model, K-ω model, Reynolds stress transport model, Algebraic Reynolds stress model Large eddy simulation model etc. have been developed to simulate the complex secondary structure in meandering channel.

K-ε model is the most common model used in CFD to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equation. The first transported variable **(K)** determines the energy in the turbulence and is called turbulence kinetic energy. The second one (ε) is the turbulence dissipation which determines the rate of dissipation of turbulence kinetic energy. This model is given to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. It focuses on the mechanism that affects the turbulent kinetic energy. It is more expensive in terms of memory. Its accuracy has been reduced for flows containing large adverse pressure gradients. It also performs poorly in variety of important cases such as unconfined flow, curved boundary layer, rotating flows, and flows in non-circular ducts.

K-ω model is a common two equation turbulence model in computational fluid dynamics which is used a closer equation of the Reynolds-averaged Navier-Stokes equation. Where **K** signifies turbulence kinetic energy and **ω** signifies specific rate of dissipation.

Large eddy simulation model (LES) model solves spatially averaged Navier stokes equation. Large eddies are directly resolved but the eddies smaller than mesh are modelled. It solves large spatial as well as smaller scale equations. It operates on the Navier-Stokes equations to reduce the range of the solution reduce the computational cost. In the present study LES model is used to simulate fluid flow in the meandering channel

CHAPTER 4

RESULT AND DISCUSSION

4. RESULTS AND DISCUSSION

4.1 Boundary Shear Stress Distribution at Different Sections along the Meander Path.

Boundary shear stress measurements have been carried out at different sections along the meander path through the cross-over. Total 13 reaches Fig.4.1 of boundary shear stress measurements have been carried out. The figures from 4.2 to 4.7 illustrate the boundary shear stress distributions across the channel bed and the side walls at the inner and outer walls for all the 13 reaches Fig.4.1

The shear stress profiles along the rigid surface of the channel are presented by showing the stress curves perpendicular to all the three sides of the channel viz. the bed, the inner wall and the outer wall. Hence the figures provide a clear demonstration about the boundary shear stress distribution throughout the channel section along the meander path.

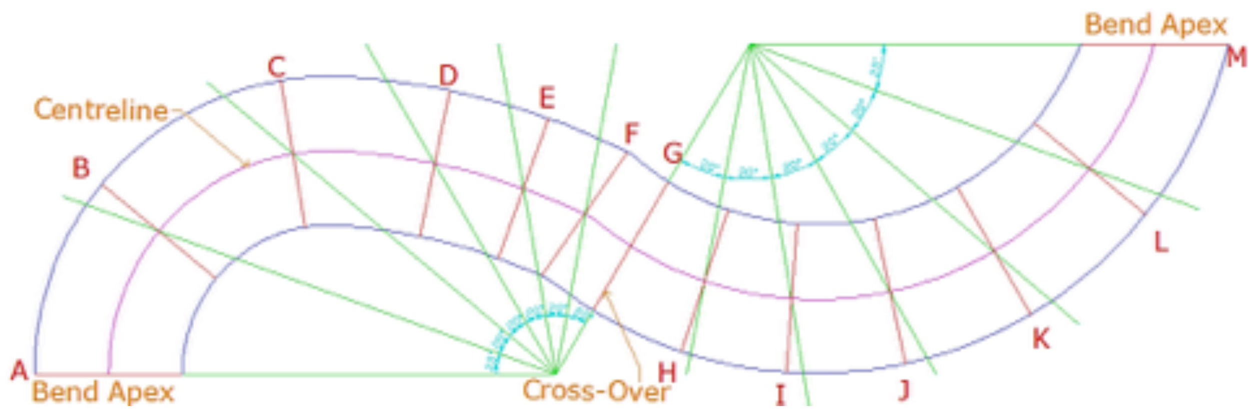

Fig.4.1. Geometry of the Meandering Path.

In the first stage, the discharge of 5.2 x 10^{-3} m^3/s is maintained for the entire experiment and the velocity data are obtained for the meandering path along the channel at every cross-section. In second stage, the discharge of 6.3 x 10^{-3} m^3/s is maintained for the entire experiment and the velocity data are obtained for the meandering path along channel at every cross-section, and in third stage, the discharge of 7.4 x 10^{-3} m^3/s is maintained for the entire experiment and the velocity data are obtained for the meandering path along the channel at every cross-section.

The boundary shear stresses are measured at inner bend and outer bend at the height of 0.2H from the bed and Bed shear stress is measured at the centre at the meander channel at bed level. The shear stress profiles along the rigid surface of the channel are represented by showing the stress curves perpendicular to all the three sides of the channel; namely the bed, the inner wall and the outer wall. Hence the figures give a clear demonstration about the boundary shear stress distribution throughout the channel section. The analysis has been performed at 13 different different sections along the path of the meandering channel, that is from one apex of the bend to another. Research has been done to analyse the variation in boundary shear stress along the wetted perimeter on the channel bed with different discharges to depict the flow characteristics of meandering channel.

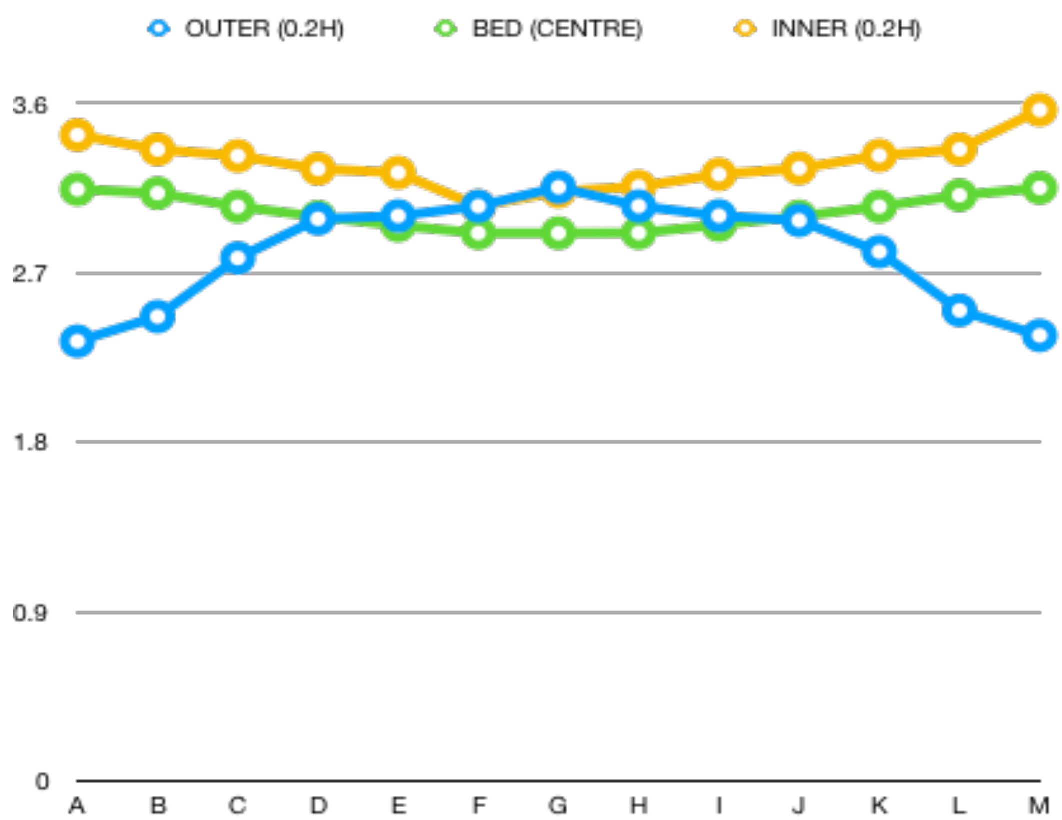

Fig.4.2. Boundary shear stresses at different sections at the discharge of 5.2 x 10^{-3} m^3/s.

From the above figure it is observed that

3. At the bend apex section A, given in figure 4.2, the shear stress carried by the inner wall is seen to be more as compared to the outer wall.

4. The highest shear stress at the inner bank of the section A is found to be around 3.4324N/m^2, while a minimum of 2.3467 N/m^2 at the outer wall. The variation of shear stress in this section is seen to be erratic.

5. The sections A to M, in Fig. 4.2, the shear stress remains higher towards the inner wall. But, in these sections, the variation of shear stress between the inner and outer walls is observed to be gradual.

6. At the cross-over section G, the variation of boundary shear stress is seen to be more or less uniform throughout the channel section. The shear stress value of inner and outer walls are close to each other and remains close to 3.14 N/m^2.

7. From the Fig. 4.2, it is observed that on moving from the initial bend apex section A; the boundary shear carried by the inner wall decreases and reaches a minimum at the cross-over. It then increases on the other bank (inner wall) while progressing towards the next bend apex section M.

8. From Fig. 4.2, it is observed that from section D to J there is a gradual variation in boundary shear stress between the inner and outer walls, with higher stress remaining towards the left bank (inner wall).

9. In the above figure from section A and M we can depict drastic variation between the boundary of the channel section, with higher shear stress lying towards the inner wall.

The shear stress profiles along the rigid surface of the channel are represented by showing the stress curves perpendicular to all the three sides of the channel; namely the bed, the inner wall and the outer wall. Hence the figures give a clear demonstration about the boundary shear stress distribution throughout the channel section. The analysis has been performed at 13 different different sections along the path of the meandering channel, that is from one apex of the bend to another. Research has been done to analyse the variation in boundary shear stress along the wetted perimeter on the channel bed with different discharges to depict the flow characteristics of meandering channel.

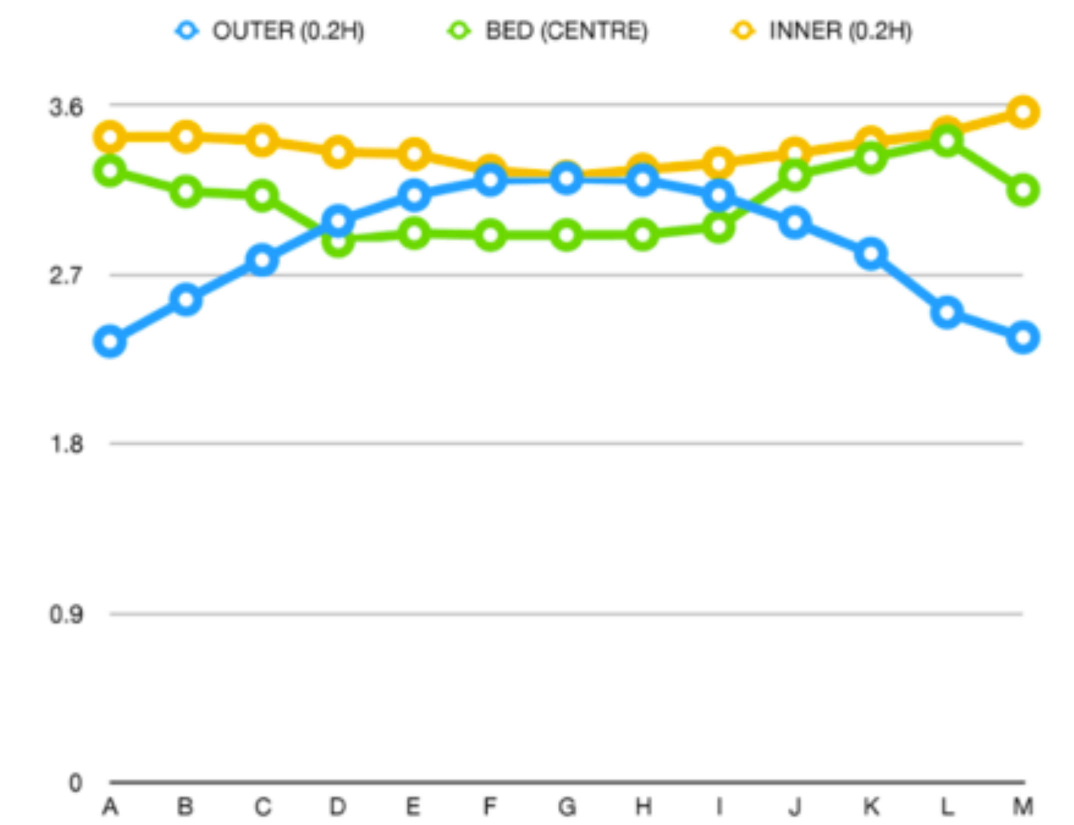

Fig.4.3. Boundary shear stresses at different sections at the discharge of 6.3 x 10^{-3} m^3/s

From the above figure it is observed that

1. At the bend apex section A, given in figure 4.1.3, the shear stress carried by the inner wall is seen to be more as compared to the outer wall.

2. The highest shear stress at the inner bank of the section A is found to be around 3.4324N/m^2, while a minimum of 2.3456 N/m^2 at the outer wall. The variation of shear stress in this section is seen to be erratic.

3. The sections A to M, in Fig. 4.1.3, the shear stress remains higher towards the inner wall. But, in these sections, the variation of shear stress between the inner and outer walls is observed to be gradual.

4. At the cross-over section G, the variation of boundary shear stress is seen to be more or less uniform throughout the channel section. The shear stress value of inner and outer walls are close to each other and remains close to 3.2 N/m^2.

5. From the Fig. 4.1.3, it is observed that on moving from the initial bend apex section A; the boundary shear carried by the inner wall decreases and reaches a minimum at the cross-over. It then increases on the other bank (inner wall) while progressing towards the next bend apex section M.

6. From Fig. 4.1.3, it is observed that from section C to K there is a gradual variation in boundary shear stress between the inner and outer walls, with higher stress remaining towards the left bank (inner wall).

7. In the above figure from section A and M we can depict drastic variation between the boundary of the channel section, with higher shear stress lying towards the inner wall.

The shear stress profiles along he rigid surface of the channel are represented by showing the stress curves perpendicular to all the three sides of the channel; namely the bed, the inner wall and the outer wall. Hence the figures give a clear demonstration about the boundary shear stress distribution throughout the channel section. The analysis has been performed at 13 different different sections along the path of the meandering channel, that is from one apex of the bend to another. Research has been done to analyse the variation in boundary shear stress along the wetted perimeter on the channel bed with different discharges to depict the flow characteristics of meandering channel.

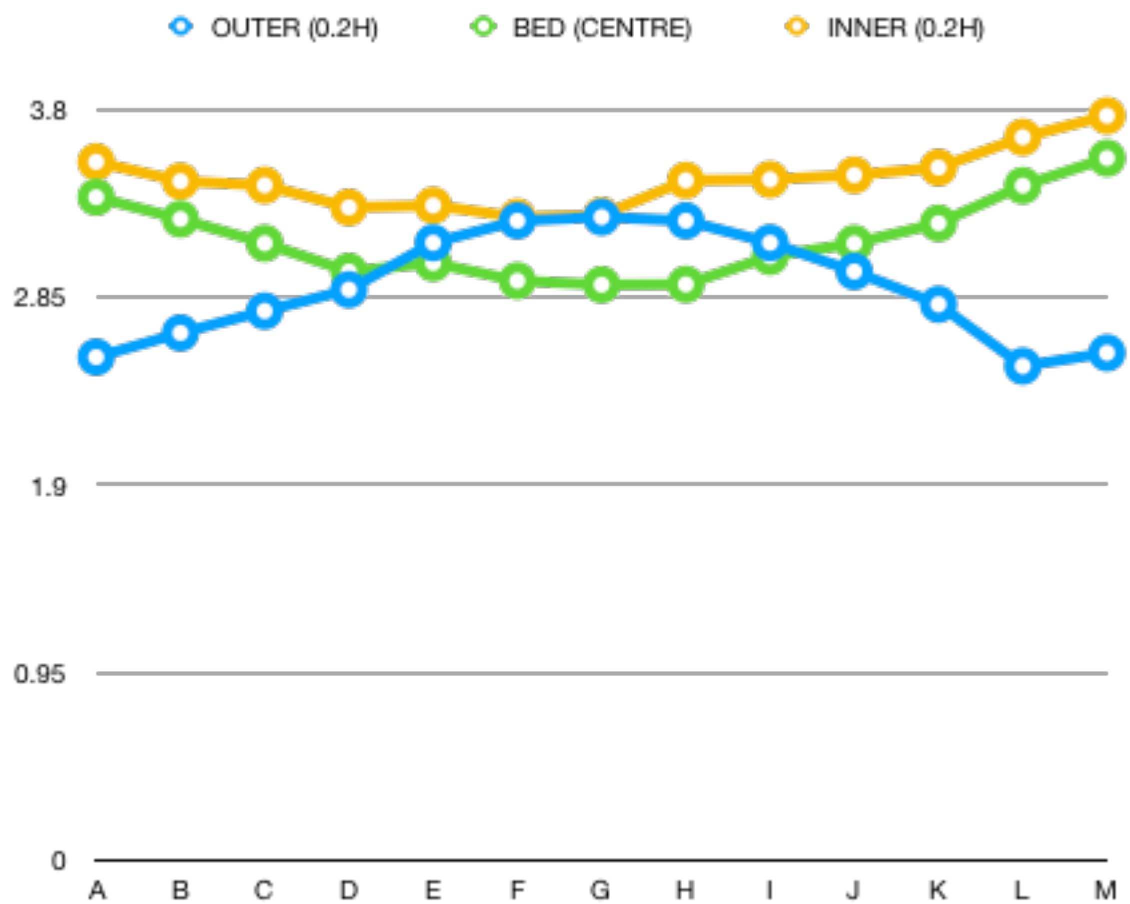

Fig.4.4. Boundary shear stresses at different sections at the discharge of 7.4 x 10^{-3} m^3/s

From the above figure it is observed that

1. At the bend apex section A, given in figure 4.4, the shear stress carried by the inner wall is seen to be more as compared to the outer wall.

2. The highest shear stress at the inner bank of the section A is found to be around 3.5324N/m^2, while a minimum of 2.5456 N/m^2 at the outer wall. The variation of shear stress in this section is seen to be erratic.

3. The sections A to M, in Fig. 4.4, the shear stress remains higher towards the inner wall. But, in these sections, the variation of shear stress between the inner and outer walls is observed to be gradual.

4. At the cross-over section G, the variation of boundary shear stress is seen to be more or less uniform throughout the channel section. The shear stress value of inner and outer walls are close to each other and remains close to 3.2 N/m^2.

5. From the Fig. 4.4, it is observed that on moving from the initial bend apex section A; the boundary shear carried by the inner wall decreases and reaches a minimum at the cross-over. It then increases on the other bank (inner wall) while progressing towards the next bend apex section M.

6. From Fig. 4.4, it is observed that from section D to J there is a gradual variation in boundary shear stress between the inner and outer walls, with higher stress remaining towards the left bank (inner wall).

7. In the above figure from section A and M we can depict drastic variation between the boundary of the channel section, with higher shear stress lying towards the inner wall

The shear stress profiles along he rigid surface of the channel are represented by showing the stress curves perpendicular to all the three sides of the channel; namely the bed, the inner wall and the outer wall. Hence the figures give a clear demonstration about the boundary shear stress distribution throughout the channel section. The analysis has been performed at 13 different different sections along the path of the meandering channel, that is from one apex of the bend to another. Research has been done to analyse the variation in boundary shear stress along the wetted perimeter on the channel bed with different discharges to depict the flow characteristics of meandering channel.

4.2 Boundary Shear Stress Distribution at Different Sections at different discharges along the Meander Path.

In the below figure Q1= 5.2 x 10^{-3} m^3/s, Q2= 6.2 x 10^{-3} m^3/s, Q3= 7.4 x 10^{-3} m^3/s. Q1, Q2, Q3 are the discharges in increasing values. Hence, it is noticed that the value of shear stress at the outer wall of the channel increases with the increase in discharge. There are total of 13 sections at the angle of 20 degree each. Section A is the bend apex of the channel which has generally small shear stress at the outer wall as compare to other sections of the channel. The shear stress at the outer wall of the channel increases with the increase in discharge and it also increases as we move from bend apex to cross-over that means from section A to section G of the outer wall and has the maximum shear stress at the cross-over G.

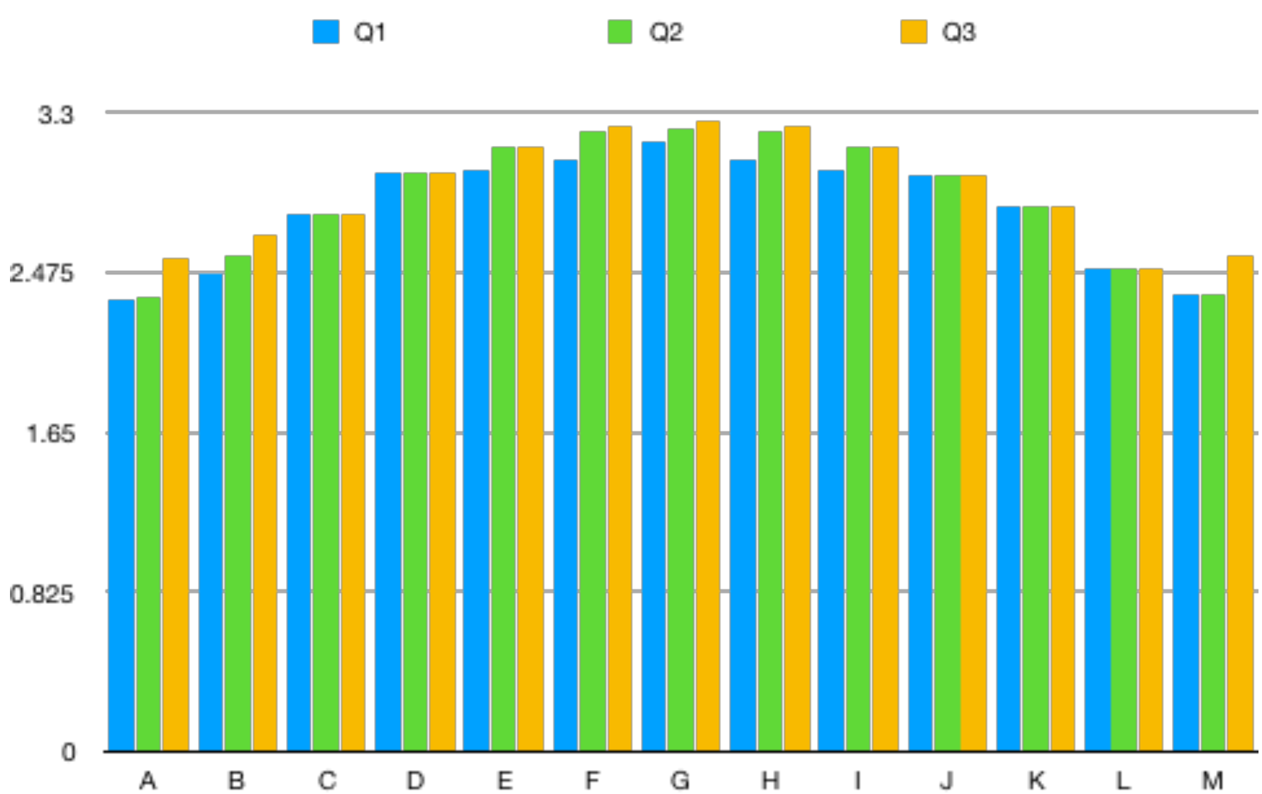

Fig 4.5 Shear Stress Distribution at outer wall of the channel at different discharges.

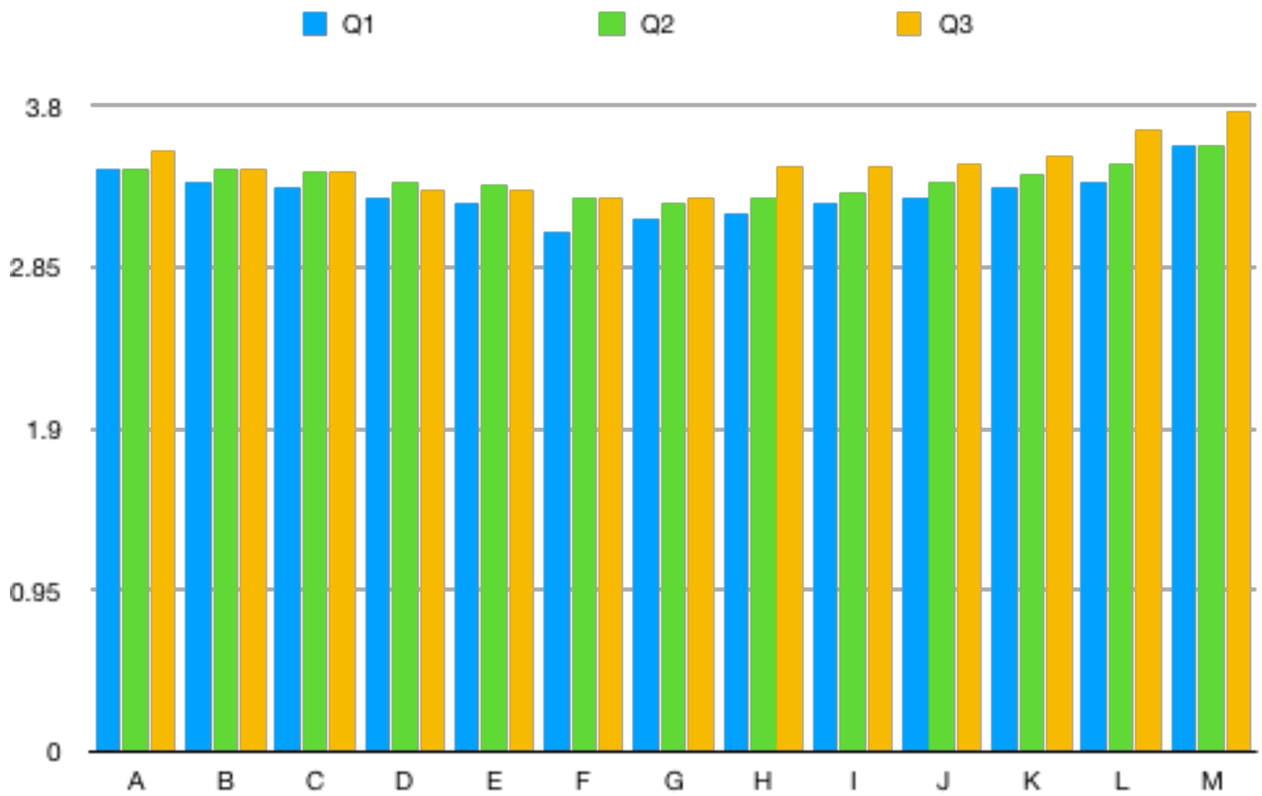

Fig 4.6 Shear Stress Distribution at inner wall of the channel at different discharges.

In the above figure Q1= 5.2 x 10^{-3} m^3/s, Q2= 6.2 x 10^{-3} m^3/s, Q3= 7.4 x 10^{-3} m^3/s. Q1, Q2, Q3 are the discharges in increasing values. Hence, it is noticed that the value of shear stress at the inner wall of the channel increases with the increase in discharge. There are total of 13 sections at the angle of 20 degree each. Section A is the bend apex of the channel which has generally larger shear stress at the inner wall as compare to other sections of the channel. The shear stress at the inner wall of the channel increases with the increase in discharge but it decreases as we move from bend apex to cross-over that means from section A to section G of the inner wall and has the minimum shear stress at the cross-over G.

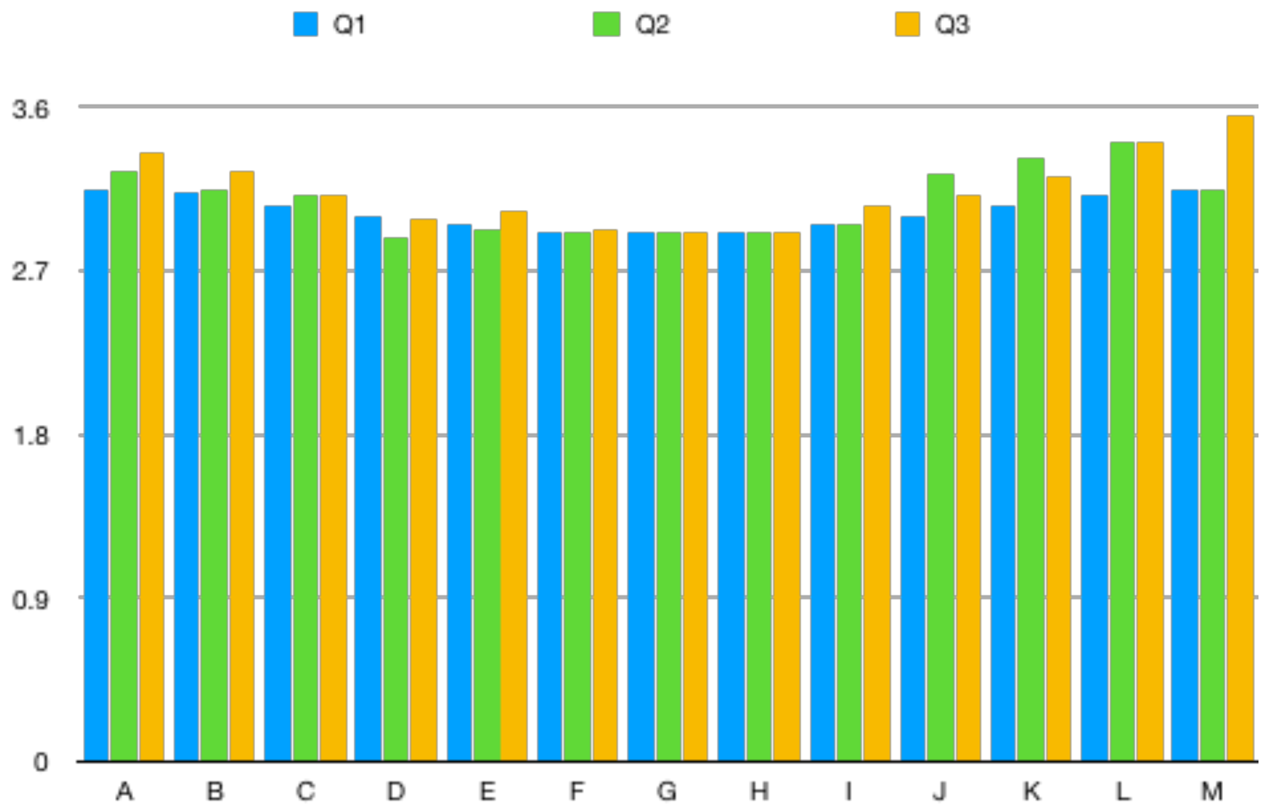

Fig 4.7 Shear Stress Distribution at inner wall of the channel at different discharges.

In the above figure Q1= 5.2 x 10^{-3} m^3/s, Q2= 6.2 x 10^{-3} m^3/s, Q3= 7.4 x 10^{-3} m^3/s. Q1, Q2, Q3 are the discharges in increasing values. Hence, it is noticed that the value of shear stress at the bed level at the centre of the channel increases with the increase in discharge. There are total of 13 sections at the angle of 20 degree each. Section A is the bend apex of the channel which has generally larger shear stress at the bed level as compare to other sections of the channel. The shear stress at the bed level of the channel increases with the increase in discharge but it decreases as we move from bend apex to cross-over and it remains nearly constant from section F to I and has the minimum shear stress at the cross-over G.

Table 4.1. Shear Stress (kN/m^2) at walls and bed level at different sections and at different discharges.

SECTION	Shear Stress at Outer Wall at Q1	Shear Stress at Inner Wall at Q1	Shear Stress at Bed level at Q1	Shear Stress at Outer wall at Q2	Shear Stress at Inner Wall at Q2	Shear Stress at Bed level at Q2	Shear Stress at Outer Wall at Q3	Shear Stress at Inner Wall at Q3	Shear Stress at Bed level at Q3
A	2.3367	3.4324	3.1436	2.3456	3.4324	3.2536	2.5456	3.5324	3.3536
B	2.4675	3.3554	3.1234	2.5678	3.4342	3.1434	2.6678	3.4342	3.2434
C	2.7785	3.3210	3.0521	2.7786	3.4141	3.1211	2.7787	3.4141	3.1211
D	2.9865	3.2521	2.9982	2.9865	3.3521	2.8782	2.9884	3.3021	2.9782
E	3.0023	3.2321	2.9512	3.1234	3.3421	2.9212	3.1234	3.3121	3.0212
F	3.0543	3.0534	2.9111	3.2045	3.2534	2.9111	3.2345	3.2534	2.9311
G	3.1548	3.1322	2.9110	3.2132	3.2222	2.9113	3.2532	3.2622	2.9113
H	3.0546	3.1587	2.9121	3.2045	3.2587	2.9131	3.2345	3.4387	2.9131
I	3.0032	3.2234	2.9543	3.1234	3.2934	2.9543	3.1234	3.4434	3.0543
J	2.9783	3.2554	3.0001	2.9783	3.3454	3.2301	2.9783	3.4654	3.1201
K	2.8123	3.3234	3.0521	2.8133	3.4034	3.3221	2.8123	3.5034	3.2221
L	2.5002	3.3565	3.1121	2.5012	3.4565	3.4121	2.5002	3.6565	3.4121
M	2.3672	3.5654	3.1512	2.3674	3.5654	3.1512	2.5672	3.7654	3.5512

Value of Q1= 5.2×10^{-3} m^3/s, Q2= 6.2×10^{-3} m^3/s, Q3= 7.4×10^{-3} m^3/s.

The table shown below shows the different values of shear stress on different surfaces of the channel i.e. on the side walls, inner and outer and on the bed surface of the channel. The value of the boundary shear stress is calculated on three different value of discharge. Based on the value of boundary shear stress calculated on different surfaces of the channel i.e. the outer wall, inner wall and the bed surface of the channel, the analysis is been made to study the behaviour of the shear stress on the surfaces on thirteen different sections of the channel at the angle of 20 degree each. from inlet to outlet of the channel. This analysis is been done at different values of discharges in increasing order.

Table 4.1 shows values of shear stresses at different sections of the channel from A to M. The value of shear stresses has been calculated at different surfaces of the meandering channel as shown in the table and graphs. Boundary shear stresses calculated at inner and outer walls and on the bed level are shown in figure. Q1, Q2, Q3 are the value of discharges in increasing order and the value of boundary shear stresses do not differ by large amount which verifies the calculation by the software. The shear stresses taken at different sections from A to M shows similar behaviour. Hence, it shows the behaviour of the channel to be same at different discharge. Boundary shear stress at the bed level have been calculated at the centre of the channel and for the outer and inner walls the shear stresses have been calculated at the height of 0.2H from the bottom of the channel. The geometry of the channel is trapezoidal with the slope of 1:1 and the bed surface of the channel is rigid and smooth.

By analysing the value of shear stress at the centre of the bed level it can be analyse that shear stress is maximum at the apex of the bed level and min at the cross over of the channel. Shear stresses at the outer surface of the channel have min value at the apex and maximum value at the cross over of the channel unlike the shear stress variation at bed level. In case of inner wall of the channel shear stress is maximum at the apex and decreases toward the cross over and becomes minimum at the cross over of the channel. All these value shows similar characteristics at different discharges which is also shown with the help of the graphs.

CHAPTER 5
CONCLUSIONS

5.1 CONCLUSIONS

Numerical analysis are carried out on a highly sinuous meander path at different reaches. The flow characteristics such as boundary shear stress distribution is investigated based on the CFD analysis of the design meandering channel, and well validated by taking different discharges.

1. At the bend apex section A, the shear stress carried by the inner wall is seen to be more as compared to the outer wall.

2. The maximum shear shear at the inner bank of the section A and is minimum at the outer wall. The variation of the shear stress in this section is seen to be erratic.

3. The section A to M, the shear stress remains higher towards the inner wall and lower towards the outer wall. But, in these sections, the variation of shear stress between the inner and outer walls is observed to be gradual.

4. At the cross-over section G, the variation of boundary shear stress is seen to be more or less uniform throughout the channel section. The shear stress value of inner and outer walls are close to each other.

5. It is observed that on moving from initial bend apex section A; the boundary shear carried by the inner wall decreases and reaches a maximum at the cross-over. It then increases on the other bank (inner wall) while progressing towards the next bend apex section M.

6. It is observed that from section D to J there is a gradual variation in boundary shear stress between the inner and outer walls, with higher stress remaining towards the left bank (inner wall).

7. It is observed that the shear stress distribution at the bed level of the channel is maximum at the apex and decreases from apex to cross-over and again increases from cross-over to apex that is from section A to G and G to M.

8. In the above figure from section A and M we can depict drastic variation between the boundary shear stress values between the inner and outer walls of the channel section, with higher shear stress lying towards the inner wall.

9. With the increase in discharge Q1, Q2, Q3 i.e. Q1= 5.2 x 10^{-3} m^3/s, Q2= 6.2 x 10^{-3} m^3/s, Q3= 7.4 x 10^{-3} m^3/s. It is noticed that the value of shear stress at inner wall, outer wall and the bed level at the centre of the channel increases with the increase in discharge. There are total of 13 sections at the angle of 20 degree each.

10. Section A is the bend apex of the channel which has generally larger shear stress at the bed level as compare to other sections of the channel. The shear stress at the bed level of the channel increases with the increase in discharge but it decreases as we move from bend apex to cross-over and it remains nearly constant from section F to I and has the minimum shear stress at the cross-over G.

11. The shear stress at the outer wall of the channel increases with the increase in discharge and it also increases as we move from bend apex to cross-over that means from section A to section G of the outer wall and has the maximum shear stress at the cross over G.

12. The shear stress at the inner wall of the channel increases with the increase in discharge but it decreases as we move from bend apex to cross-over that means from section A to section G of the inner wall and has the minimum shear stress at the cross over G.

5.2 SCOPE FOR FUTURE RESEARCH

The-present work gives a vast scope for further investigations of various aspects of a meandering river/channel. The-present research is limited to one particular angle and analysis has been done at 13 sections only . The research can be continued for different discharges at different angles with various number of sections to get an overall depiction about the flow characteristics. The future scope of the present work can be summarised as:

1. This numerical modelling can be done by meandering channel of different sinuosity.
2. Keeping the sinuosity same, the bed may be considered as rough by taking different materials of different roughness values as bed material.
3. Number of sections considered for analysis can be changes to depict the behaviour of the meander channel in more holistic way.

CHAPTER 6
REFERENCES

6. REFERENCES

1. Absi, R. (2011). "An ordinary differential equation for velocity distribution and dip-phenomenon in open channel flows" *Journal of Hydraulic Research*, IAHR, Taylor and Francis, Vol. 49, N° 1, pp. 82-89

2. Ansari.K.,Morvan.H.P. andHargreaves.D.M. (2011),"Numerical Investigation into Secondary Currents and Wall Shear in Trapezoidal channels." *Journal of HydraulicEngineering (ASCE)2011*; Vol.137 (4):432-440.

3. ArpanPradhan, Saine S. Dash, K.K.Khatua, "Water Surface Profile along a meander path of a Sinuous Channel", IOSR Journal of Mechanical and Civil Engineering (IOSR- JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, PP 48-52.

4. Arpan Pradhan, Kishanjit K. Khatua and Debashish Khuntia, "Study of Variation in Velocity Profile along a 120° Meandering Path" INROADS- An International Journal of Jaipur National University Year: 2014, Volume: 3, Issue: 1s, pp 157-160 Print ISSN : 2277-4904. Online ISSN: 2277-4912.

5. Arpan Pradhan, K. K. Khatua, Saine S. Dash, "Boundary Shear Force Distribution along different reaches of a Highly Meandering Channel" International Journal of Scientific Engineering and Technology, Special Issue: HYDRO-2014 International, 19th International Conference on Hydraulics, Water Resources and Environmental Engineering, pp 202-207, ISSN : 2277-1581

6. Arpan Pradhan, Kishanjit. K. Khatua, Saine S. Dash, "Distribution of Depth-Averaged Velocity along a Highly Sinuous Channel", Elsevier Aquatic Procedia, ICWRCOE 2015, Vol 4, pp 805-811.

7. Baghalian, S.,Bonakdari, H., Nazari, F., Fazli, M.(2012), "Closed-Form Solution for Flow Field in Curved Channels in Comparison with Experimental and Numerical Analyses and Artificial Neural Network". Engineering Applications of Computational *Fluid Mechanics* Vo.6, No.4, pp. 514-526.

8. Bathurst, J. C., Hey, R. D., & Thorne, C. R. (1979). "Secondary flow and shear stress at

9. Bodnar T, Prihoda. Numerical simulation of turbulent free-surface flow in curved Channel Flow, turbulence and combustion, 76 (2006):pp. 429-442

10. Booij R. Measurements and large eddy simulations of the flows in some curved flumes, J. Turbulence, 4 (2003):pp. 1-17

11. Bousmar D, Zech Y. Momentum transfer for practical flow computation in compound channels, Journal of Hydraulic Engineering, 125 (1999):pp. 696-706

12. Bousmar D, Zech Y. Periodical turbulent structures in compound channels, River Flow International Conference on Fluvial Hydraulics, Louvain-la-Neuve, Belgium, 2002:pp.

13. Boussinesq, J. (1868). Mémoiresurl'influence des frottementsdans les movement reguliers des fluids. J. Math. Pures Appl. (2me sér.), 13, 377-424.

14. Chang, H. H. (1984), "Variation of flow resistance through curved channels", *Journal of Hydr. Engrg.,*ASCE, 110(12), 1772–1782.

15. Chow, V. T. (1959), "Open Channel Hydraulics", McGraw-Hill Book Co, New York.

16. Dash, S. S. (2013), "Stage-Discharge Modelling of Meandering Channel". Thesis Presented to the National Institute of Technology, Rourkela, in partial fulfilment of the requirements for the Degree of Doctor of philosophy.

17. Khatua, K. K. (2008), "Interaction of flow and estimation of discharge in two stage meandering compound channels". Thesis Presented to the National Institute of Technology, Rourkela, in partial fulfilment of the requirements for the Degree of Doctor of philosophy.

18. Knight, D. W. (1981). "Boundary shear in smooth and rough channels." *J. Hydraul. Div., Am. Soc. Civ. Eng.,* 107(7), 839–851.

19. Mohanta.A(2014)" Flow Modelling of Non Prismatic compound channel By Using C.F.D"Thesis Presented to the National Institute of Technology, Rourkela, in partial fulfilment of the requirements for the Master of technology.

21. Patnaik, M. (2013), "Boundary Shear Stress Distribution in Meandering Channels". Thesis

Presented to the National Institute of Technology, Rourkela, in partial fulfilment of the requirements for the Degree of Doctor of philosophy.

22. Patra, K.C., and Kar, S.K., Bhattacharya, A.K. (2004). “Flow and Velocity Distribution in Meandering Compound Channels.”*, Journal of Hydraulic Engineering*, *ASCE*, Vol. 130, No. 5. 398-411.

23. Saine S. Dash, K.K.Khatua, P.KMohanty(2013), “Energy loss for a highly Meandering open Channel Flow”, *Res. J. Engineering Sci.,* Vol. 2(4), 22-27, April (2013).