MODELING FLUID FLOW IN OPEN CHANNEL WITH CIRCULAR CROSS – SECTION

DADDY PETER TSOMBE

MASTER OF SCIENCE

(Applied Mathematics)

JOMO KENYATTA UNIVERSITY OF

AGRICULTURE AND TECHNOLOGY

2011

Modeling fluid flow in open channel

with circular cross – section

Daddy Peter Tsombe

A thesis submitted in partial fulfillment for the degree of Master of Science in Applied Mathematics in the Jomo Kenyatta University of Agriculture and Technology

2011

DECLARATION

This thesis is my original work and has not been presented for a degree in any other University.

Signature…………………………… Date……………………………………...

Daddy Peter Tsombe

This thesis has been submitted with our approval as University Supervisors.

Signature…………………………… Date…………………………………......

Prof. Mathew Kinyanjui

JKUAT Kenya

Signature…………………………… Date…………………………………......

Prof. Jackson K. Kwanza

JKUAT Kenya

DEDICATION

This thesis is dedicated to my wife Isabella Achieng Odhiambo, dad Moses Odero Tsombe and mum Florence Caren Akeyo Tsombe.

ACKNOWLEDGEMENTS

Firstly, I would like to thank Professor Mathew Kinyanjui and Professor Jackson Kwanza for their help, support and valuable guidance throughout my research. They always showed me a high-quality way to do research and encouraged me to do my best. Their supervision, support and patience were constant throughout the period of my study. Their keen interest and understanding of the topic was inspiring and led to many fruitful discussions. Special thanks also for having had faith in me and in my abilities, it made me feel more confident and positive. Professor Kinyanjui, thank you for always keeping me awake and on toes.

Many thanks to all the postgraduate students at the Department of Pure and Applied Mathematics of Jomo Kenyatta University of Agriculture and Technology. Their reciprocal support not only during the exciting and productive but also the demanding phases of this project has been very important. I wish to particularly thank Dr. Okelo, Dr. Mwenda and Mr. Kang'ethe, who were always helpful whenever I needed help. Mr Kang'ethe, thank you for your invaluable assistance especially for working tirelessly to write and run the computer programme for the problems solved in this thesis.

I wish to thank the staff of the PAM department who always made me feel welcome. Working there afforded me the opportunity to meet very good researchers with whom we shared very happy moments that I will cherish forever. I am also grateful to all my

friends and all those nice people for their assistance and encouragements throughout several challenging times during the last years.

To my wife Isabella, many thanks indeed for believing in me. You gave me the extra strength, motivation and support necessary to get things done. To my family Beatrice, Beryl, Henry, Elizabeth, Francis, dad Moses Odero Tsombe and mum Florence Caren Akeyo Tsombe (Nyar Sondu), thank you for your infinite love and encouragement. You have always been a continual source of support. Finally, thanks to God the almighty for His infinite mercies.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF APPENDICES

LIST OF SYMBOLS

ROMAN SYMBOLS

Symbol	**Meaning**
$\boldsymbol{A}$	Cross- sectional area of flow (m^2)
C	Resistance coefficient of the flow (Chezy coefficient)
$\boldsymbol{D}$	Hydraulic depth (m)
$\boldsymbol{Fr}$	Froude number (dimensionless)
$\boldsymbol{H}$	Total energy at a cross-section
$\boldsymbol{L}$	Length of the channel (m)
$\boldsymbol{P}$	Wetted perimeter of the channel cross section (m)
$\boldsymbol{Q}$	Discharge (m^3s^{-1})
$\boldsymbol{R}$	Hydraulic Radius (m)
$\boldsymbol{Re}$	Reynolds number (dimensionless)
$\boldsymbol{S}$	Slope of the channel bottom
$\boldsymbol{S_f}$	Friction slope
$\boldsymbol{T}$	Top width of the free surface (m)
$\boldsymbol{V}$	Mean velocity of flow (m/s)
$\boldsymbol{g}$	Acceleration due to gravity (ms^{-2})
$\boldsymbol{m}$	Dimensions of length
$\boldsymbol{n}$	The Manning coefficient of roughness ($sm^{-1/3}$)
$\boldsymbol{q}$	Lateral/uniform inflow (m^2s^{-1})
$\boldsymbol{r}$	Radius of the conduit

$\boldsymbol{t}$	Time (s)
$\boldsymbol{x}$	Distance along the main flow direction (m).
$\boldsymbol{y}$	Depth of the flow (m)
$\boldsymbol{z}$	Co-ordinate direction, bed level relative to a datum

GREEK SYMBOLS

$\boldsymbol{\alpha}$	Energy coefficient
$\boldsymbol{\nu}$	Kinematic viscosity (m^2/s)
$\boldsymbol{\gamma}$	Fluid specific weight
$\boldsymbol{\beta}$	Longitudinal slope angle
$\boldsymbol{\theta}$	Angle between two radii of a uniform circular conduit
$\boldsymbol{\tau}$	Shear stresses

ABSTRACT

Flow in a closed conduit is regarded as open channel flow, if it has a free surface. This study considers unsteady non-uniform open channel flow in a closed conduit with circular cross-section. We investigate the effects of the flow depth, the cross section area of flow, channel radius, slope of the channel, roughness coefficient and energy coefficient on the flow velocity as well as the depth at which flow velocity is maximum. The finite difference approximation method is used to solve the governing equations because of its accuracy, stability and convergence followed by a graphical presentation of the results. It is found that for a given flow area, the velocity of flow increases with increasing depth and that the velocity is maximum slightly below the free surface. Moreover, increase in the slope of the channel and energy coefficient leads to an increase in flow velocity whereas increase in roughness coefficient, flow depth, radius of the conduit and area of flow leads to a decrease in flow velocity.

CHAPTER ONE

1.0 INTRODUCTION

Water flows more rapidly on steeper slope but for a constant slope, the velocity reaches a steady value when the gravitational force is equal to the resistance to flow. Over the years, man has endeavoured to direct water to the desired areas such as farms, where it is used for irrigation. He has also tried to draw water from storage sites such as reservoirs, dams and lakes. To achieve this objective, he has constructed open channels which are physical systems in which water flows with a free surface.

The cross-section of these channels may be open or closed at the top. The structures with closed tops are referred to as *closed conduits* while those with open tops are called *open channels*. For example, tunnels and pipes are closed conduits whereas rivers and streams are open channels. Many procedures have been developed over the years for the hydraulic design of open channel sections. This chapter begins with a review of the terminologies used in open-channel hydraulics followed by a review of literature related to this work. The problem considered in this study is also presented. Towards the end of the chapter, the objectives are stated.

1.1 OPEN CHANNEL

An open channel is a conduit for flow which has a free surface, i.e., a boundary exposed to the atmosphere. The free surface is an interface between two fluids of different densities. In the case of the atmosphere, the density of air is much lower than the density for a liquid such as water. In open channel flow, liquid flows with a free surface subjected to atmospheric pressure and the motion is usually caused by gravitational effects.

Open channel flows are almost always turbulent and unaffected by surface tension. As defined above, open channel flow include flow occurring in channels ranging from small streams flowing across a field to gutters along residential streets to partially filled sewers carrying waste water to irrigation channels carrying water.

1.2 KINDS OF OPEN CHANNEL

Classified according to its origin, a channel may be either natural or artificial. Artificial channels are those constructed or developed by human effort. They are also known as man-made channels and include irrigation canals, drainage ditches, sewers, power canals as well as model channels that are built in the laboratory for testing purposes. They are normally of regular cross-section and bed slope, and as such are termed prismatic channels. Their construction materials are varied, but commonly used materials include concrete, steel and earth. The surface roughness characteristics of these materials are normally well defined. The hydraulic properties of such channels can be either controlled to the extent desired or designed to meet given requirements. The applications of hydraulic theories to artificial channels will, therefore, produce results fairly close to actual conditions and, hence, are reasonably accurate for practical design purposes.

Natural channels on the other hand are normally very irregular in shape, and their materials are diverse. The surface roughness of natural channels changes with time, distance and water surface elevation. Therefore it is more difficult to apply hydraulic theory to natural channels and obtain satisfactory results since the hydraulic properties of natural channels are very irregular.

1.3 CHANNEL GEOMETRY

A channel built with unvarying cross-section and constant bottom slope is called a prismatic channel. Otherwise the channel is called non prismatic. Unless specifically indicated, the channel described in this work is prismatic and the term channel section as used in this work refers to the cross section of a channel taken normal to the direction of flow.

1.4 FLOW CLASSIFICATION

It is possible to classify the type of flow occurring in an open channel on the basis of many different criteria. One of the primary criteria for classification is the variation of depth of flow y in time t and space x. If time is the criterion, then a flow can be classified as being either *steady*, which implies that the depth of flow does not change with time ($\partial y/\partial t = 0$), or *unsteady*, which implies that the depth does change with time ($\partial y/\partial t \neq 0$). If space is used as the classification criterion, then a flow can be classified as *uniform* if the depth of flow does not vary with distance ($\partial y/\partial x = 0$) or as *non uniform* if the depth varies with distance ($\partial y/\partial x \neq 0$).

Non uniform flow, also termed *varied flow* is further classified as being either *rapidly varied* - the depth of flow changes rapidly over a relatively short distance such as is the case with a hydraulic jump - or *gradually varied* - the depth of flow changes rather slowly with distance such as is the case of a reservoir behind a dam. The major classifications applied to open channels are as follows: *Steady uniform flow,* in which the depth is constant, both with time and distance. This constitutes the fundamental type of flow in an open channel in which the gravity forces are in equilibrium with the resistance forces. *Steady non-uniform flow,* in which the depth varies with distance but not with time. The flow may be either gradually varied or rapidly varied. *Unsteady uniform flow,* in which the flow variables vary with time but not with distance. Although conceptually this flow is possible, from a practical viewpoint such a flow is nearly impossible. *Unsteady non-uniform flow,* in which the flow variables vary with time and distance along the channel.

1.5 STATE OF FLOW

The state or behavior of open channel flow is governed by the effects of viscosity and gravity relative to the inertia forces of the flow. The surface tension of water may affect the behaviour of flow under certain circumstances, but it does not play a significant role in most open channel problems. Depending on the effect of viscosity relative to inertia, the flow may be laminar, turbulent or transitional.

The flow is laminar if the viscous forces are so strong relative to the inertial forces that viscosity plays a significant part in determining flow behavior. In laminar flow, the water particles appear to move in definite smooth paths, or streamlines, and infinitesimally thin layers of fluid seem to slide over adjacent layers. The flow is turbulent if the viscous forces are weak relative to the inertial forces. In turbulent flow, the water particles move in irregular paths which are neither smooth nor fixed but which in the aggregate still represent the forward motion of the entire stream. A transitional flow is one which can be classified as neither laminar nor turbulent.

The effect of viscosity relative to inertia can be represented by a dimensionless parameter known as Reynolds number, defined as: $Re = VL \, / \, \nu$ where V is the mean velocity of flow, the characteristic length L is taken to be the hydraulic radius R which is the ratio of flow area A to the wetted perimeter P and ν is the kinematic viscosity. When the hydraulic radius is taken as the characteristic length, the flow in a pipe changes from laminar to turbulent in the range of Re between the critical value of 500 and a value that may be as high as 12,500. Then if: $Re \leq 500$ the flow is laminar, $500 \leq Re \leq 12500$ the flow is transitional and $12500 \leq Re$ the flow is turbulent. In most open channels, laminar flow occurs very rarely since most open-channel flows involve water (which has a fairly small viscosity) and have relatively large characteristic lengths. Hence it is uncommon to have laminar open channel flows. Laminar open channel flow is known to exist, however, usually where thin sheets of water flow over the ground or where it is created deliberately in model testing channels.

The importance of gravity as a driving force in open channel drainage systems makes its effect on the state of flow a major factor for evaluation. This can be done using a dimensionless parameter known as the Froude Number, represented by a ratio of inertial forces to gravity forces and expressed mathematically as: $Fr = V / \sqrt{gL}$ where V is the mean velocity of flow, the characteristic length L is taken to be the hydraulic depth D, which is the ratio of flow area A to the width of the free surface T and g is the acceleration due to gravity. When the Fr value equals 1, inertial forces and gravity forces are balanced and the open channel exhibits critical flow. If Fr values are less than 1, or $V < \sqrt{gD}$, gravity forces dominate and the open channel is said to be operating in the sub-critical range of flow. This is sometimes called tranquil flow and is characterized as relatively deep, low velocity flow with respect to critical flow conditions. Depth of flow is controlled at a downstream location. If Fr values are greater than 1, or $V > \sqrt{gD}$, inertial forces dominate and the open channel is said to be operating in the super-critical range of flow. This is also called rapid, shooting or torrential flow and is characterized as relatively shallow, high velocity flow with respect to critical flow condition. Depth of flow can be controlled at an upstream location.

Open-channel flows involve a free surface that can deform from its undisturbed relatively flat configuration to form waves. The character of an open channel flow may depend strongly on how fast the fluid is flowing relative to how fast a typical wave moves relative to the fluid. The Froude number, $Fr = V/(gy)^{1/2}$, where y is the fluid depth, is the dimensionless parameter that describes this behaviour.

1.6 NEWTONIAN FLUIDS

In a Newtonian fluid, the relation between the shear stress and the strain rate is linear (and if one were to plot this relationship, it would pass through the origin), the constant of proportionality being the coefficient of viscosity.

For a Newtonian fluid, the viscosity, by definition, depends only on temperature and pressure (and also the chemical composition of the fluid if the fluid is not a pure substance), not on the forces acting upon it.

In common terms, this means the fluid continues to flow, regardless of the forces acting on it. For example, water is Newtonian, because it continues to exemplify fluid properties no matter how fast it is stirred or mixed.

1.7 NON-NEWTONIAN FLUIDS

In a non-Newtonian fluid, the relation between the shear stress and the strain rate is nonlinear, and can even be time-dependent. Therefore a constant coefficient of viscosity cannot be defined.

A non-Newtonian fluid is a fluid whose flow properties are not described by a single constant value of viscosity. Many polymer solutions and molten polymers are non-Newtonian fluids, as are many commonly found substances such as starch suspensions, paint and blood.

1.8 LITERATURE REVIEW

Open channel flow is a familiar sight, whether in a natural channel like that of a river, or an artificial channel like that of an irrigation ditch. The principal forces at work are those of inertia, gravity and viscosity, each of which plays an important role. Open channels has been a subject of study for a long time with the Chézy equation as one of the earliest semiempirical uniform flow equations developed for computation of average velocity of a uniform flow, Henderson (1966). The formula that is used most in open channels is Manning formula, developed empirically (Bilgil, 1998) through the studies he performed Manning (1895) The Manning equation has proved to be very reliable in practice. Studies in open channel flows have to take into account the coefficient of roughness, called the *Manning coefficient.* The Manning coefficient takes into account the bed materials, degree of channel irregularity, variation in shape and size of the channel and relative effect of channel obstruction, vegetation growing in the channel and meandering, Chadwick and Morfet (1993). This makes the Manning equation more desirable for the design of open channels.

Chow (1959) studied open channel flows and developed many relationships such as velocity formula for open channel flow. Akbari and Firoozi (2010) investigated two different numerical methods, namely; Preissmann and Lax diffusive schemes for numerical solution of Saint-Venant equations that govern the propagation of flood wave, in natural rivers, with the objective of the better understanding of this propagation process. The results showed that the hydraulic parameters play important role in the flood wave propagation.

Moshirvaziri *et al* (2010) examined numerically, the nature of pollutant connectivity between unsealed forest roads and adjacent nearby streams in terms of spatial and temporal patterns of runoff generation, erosion, and sediment transport with an aim of improving the ability to scale-up the impacts of forest roads on catchment water quality in future works. They considered the relative effects of rainfall intensity and duration, surface roughness, infiltration rate, macropore flow and sediment detachment and transport with an objective of identifying the dominant processes and parameters that affect the degree of pollutant connectivity between roads and streams. The St. Venant equations were applied to extract a two-dimensional diffusion wave model with variable conductivity and diffusivity to represent the behaviour of flow dynamics mathematically.

Chamka (2010) considered coupled heat and mass transfer by mixed convection for a non-Newtonian power-law fluid flow over a permeable wedge embedded in a fluid-saturated porous medium under variable wall temperature and concentration and in the presence of wall transpiration and heat generation or absorption effects. The obtained results were illustrated graphically and the physical aspects of the problem discussed. The results focused on the effects of the buoyancy ratio, power-law fluid index, mixed convection parameter, wall temperature or concentration, Lewis number, free stream velocity exponent, transpiration parameter and the heat generation or absorption parameter on the reduced local Nusselt and Sherwood numbers.

Kwanza *et al* (2007) analyzed the effects of channel width, slope of the channel and lateral discharge on fluid velocity and channel discharge for both rectangular and trapezoidal channels. They noted that the discharge increases as the specified parameters are varied upwards. Chagas and Souza (2005) sought to provide solution of Saint Venant's Equation to study flood in rivers through Numerical methods. They used a discretization, for the equations that govern the propagation of a flood wave, in natural rivers, with the objective of a better understanding of this propagation process. Their results showed that the hydraulic parameters play important role in the propagation of a flood wave.

Khan (2000) studied open channel flow over an initially dry bed with the aim of better understanding flow over islands during rising flood stage, flow downstream of the hydraulic structures (such as dams and gates) during intermittent release of water, and flood wave, either due to natural causes or sudden failure of a hydraulic structure, over an initially dry bed.

Tuitoek and Hicks (2001) modeled unsteady flow in compound channels with an aim of controlling floods. They developed a model based on the St. Venant equations of flow with incorporation of terms to account for the momentum transfer phenomenon, to route unsteady flow in compound channels. For the main channel, the full dynamic equations were used while in the floodplains, a diffusive model was used. Both included mass and momentum transfer terms. The resulting equations called CCDG 1-D model were solved by the characteristic-dissipative-Galerkin (CDG) finite element method. Results from the CCDG 1-D model were compared to observed experimental data. The unsteady results also showed that the CCDG 1-D model showed improvement on discharge prediction when the apparent shear was included in the model.

Sinha and Meena (1982) investigated the development of the laminar flow of a viscous incompressible fluid from the entry to the fully developed situation in a straight circular pipe. They observed that the velocity increases more rapidly during the initial development of the flow in comparison to the downstream flow. It was also observed that during the initial stages of the development of the flow, the rate of increase in stream wise velocity is larger and consequently the pressure drop is larger in comparison with their values further downstream. This is because the retarded fluid particles in the boundary layer are pushed towards the core more rapidly near the entry where the boundary layer is thinner as compared to the downstream region.

Crossley (1999) investigated strategies developed for the Euler's equations for application to the Saint Venant equations of open channel flow in order to reduce run times and improve the quality of solutions in the regions of discontinuities. Nalluri and Adepoju (1985) analysed experimental data on resistance to flow in smooth channels of circular cross-section. The results of the tests showed that the measured friction factors are larger than those for a pipe of equivalent diameter (D - 4R). Through the analysis of the data for the range of flow depths $0 < y < 0.85D$, the geometric parameter y/P was found more appropriately representing the shape effects on resistance to flow in circular channels than using a single parameter like the hydraulic mean radius, *R*.

1.9 STATEMENT OF THE RESEARCH PROBLEM

In this study, unsteady non-uniform flow of Newtonian fluid through a partly full prismatic circular pipe is considered. The effect of velocity and depth as they vary from one point to another is sought in order to understand the relation between the two. The effects of the cross section area of flow, channel radius, slope of the channel, roughness coefficient, energy coefficient and wetted perimeter on the flow velocity is sought whereby a parameter will be varied one at a time holding all the other parameters constant.

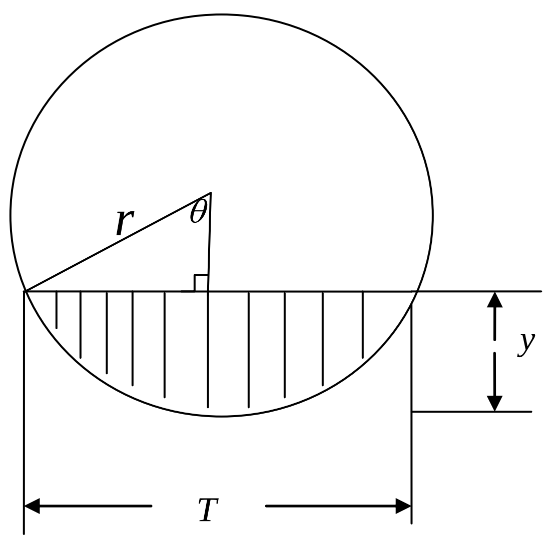


Figure 1.1: Cross-Section of a circular conduit

From figure 1.1

i) The wetted perimeter, is a section of the circumference of the conduit which is in contact with the fluid. It is obtained from:

$$P = \frac{2\theta}{2\pi} 2\pi r = 2r\theta \qquad (1.1)$$

ii) The width of the free surface is obtained from:

$$\sin\theta = \frac{T/2}{r} \Rightarrow \frac{T}{2} = r\sin\theta \ \Rightarrow T = 2r\sin\theta \qquad (1.2)$$

iii) The cross sectional area of flow (i.e. area of the shaded segment) is obtained from:

$$A = \frac{2\theta}{2\pi}\pi r^2 - \frac{1}{2}r^2 \sin 2\theta = r^2\,(\theta - \sin\theta\cos\theta) \qquad (1.3)$$

The above equations are utilized in the determination of the hydraulic depth and hydraulic radius when solving the specific equations (3.10)and (3.13) respectively.

1.10 JUSTIFICATION

From the viewpoint of quantity and quality, water resource projects are of paramount importance to the maintenance and progress of civilization since one of the primary requirements for the development, maintenance, and advancement of civilization is access to a plentiful and economic supply of water. The knowledge of open channel hydraulics is essential to the designers of open channel projects as it provides guidelines for the hydraulic analysis and design of open channel flows.

It also provides an understanding into the propagation of flood wave in natural rivers, originating from torrential rains or of breaking of a control structure with a view of improving the design of the control structures for floods. In addition, this study is beneficial in the construction of canals and waterways to drain water from stagnation sites such as pools and dams to desired areas such as farms, where it is used for irrigation.

Most studies on open channel flow have focused on rectangular and trapezoidal shaped open channels, with flow in circular shaped channels receiving little attention. The present study seeks to investigate the effects of the various flow parameters on the flow velocity for open channel flow in a closed conduit with a circular cross-section.

The findings of this study will also be highly applicable in water mills where large volume of high velocity water is required to turn large turbines that drive mechanical processes such as flour or textile production or metal shaping and in the generation of hydro-electric power. It will also provide guidelines to designers of open channel projects such as water mills where large volumes of high velocity water is required to drive mechanical processes.

1.11 OBJECTIVES AND HYPOTHESIS

1.11.1 Objectives

The objectives of this study are:

i) To investigate the effects of the cross-sectional area of flow on the flow velocity.

ii) To investigate the effects of channel radius, roughness coefficient, energy coefficient and slope of the channel on the flow velocity.

iii) To investigate the flow depth at which the flow velocity is maximum.

1.11.2 Null Hypothesis

Flow velocity decreases with increase in the depth of flow. Increase in channel radius, cross-section area of flow, slope of the channel and lateral discharge leads to a decrease in the flow velocity which in turn leads to a decrease in discharge.

CHAPTER TWO

2.0 INTRODUCTION

This chapter begins by a review of some uniform-flow formulas that have been developed over the years, including the Chezy and Manning formulae. The Saint Venant equations which govern open channel flow are thereafter outlined. Towards the end of the chapter, the method of solution is discussed.

2.1 VELOCITY OF A UNIFORM FLOW

Although steady uniform flow is rare in drainage facilities, it is practical in many cases to assume that steady uniform flow occurs in appropriate segments of an open channel system. The results obtained from calculations based on this assumption will be approximate and general, but can still provide satisfactory solutions for many practical problems.

For uniform flow conditions, the energy grade line is parallel to the hydraulic grade line, which is parallel to the channel bottom. Thus, for uniform flow, the slope of the channel bottom becomes an adequate basis for the determination of friction losses. During uniform flow, no conversions occur between kinetic and potential forms of energy.

There have been developed and published a large number of uniform-flow formulas, but none of these formulas meets the qualifications of a good formula, which should take equal account of the following variables: the cross-sectional area of flow, the mean velocity, the wetted perimeter, the hydraulic radius, the channel roughness, and many other factors.

Theoretical uniform-flow formulas have also been derived on the basis of a theoretical velocity distribution across the channel section. The best known and most widely used formulas are the Chezy and Manning formulas.

2.2 THE CHEZY AND MANNING FORMULAE

2.1.1 The Chezy formula

This formula can be derived by considering two hypotheses (Çeçen, 1982). The first hypothesis, which was proposed by Chezy, expresses that friction force on the wall is proportional to square of the velocity.

$$F_R \propto V^2$$

$$\Rightarrow F_R = KV^2 \tag{2.1}$$

where K is a constant of proportionality.

The surface of contact of the flow with the stream bed is equal to the product of the wetted perimeter and the length of the channel reach, or PL. Then for a reach of length L with a wetted perimeter P, the total force resisting the flow is:

$$F_R = LPKV^2 \tag{2.2}$$

The second hypothesis, which was proposed by Brahms, is the fundamental principle of uniform flow. It shows that "$W \sin\beta$" weight force which provides the flow of the liquid in general sense is equal to total friction force.

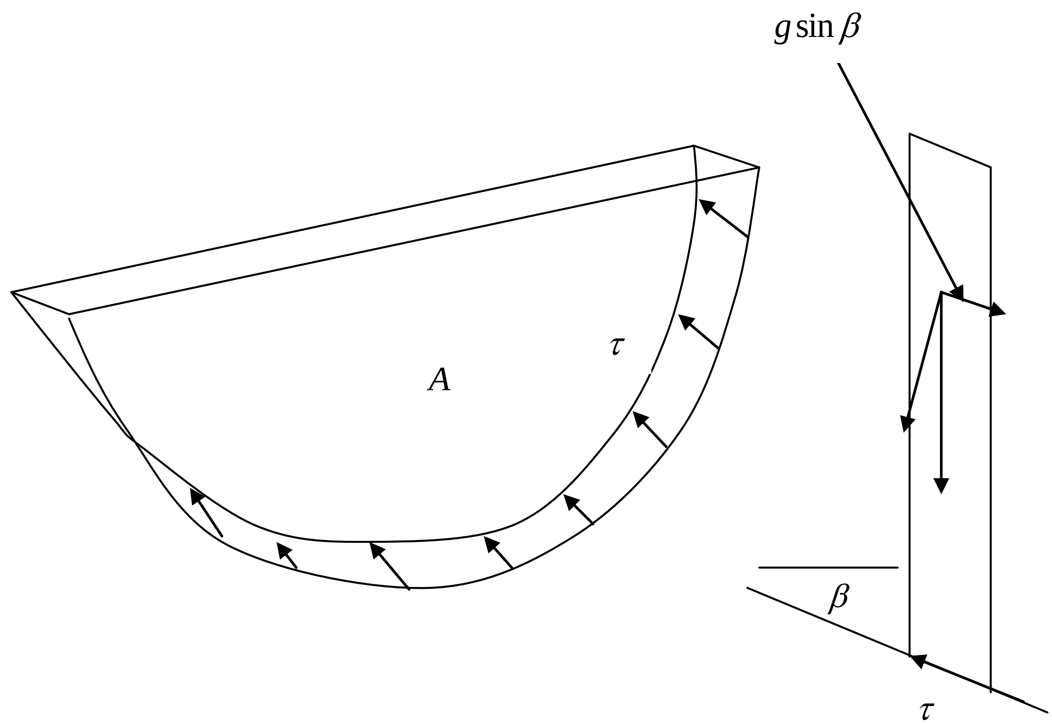


Figure 2.1: Slice of uniform channel flow showing shear forces and body forces per unit mass acting

$$F_m = W \sin\beta = \gamma AL \sin\beta \equiv \gamma ALS \tag{2.3}$$

Where W is weight of fluid within control volume, γ is fluid specific weight and β is the angle of inclination of the channel, assumed positive downwards. Setting the force causing motion, equation (2.3), equal to the force resisting motion, equation (2.2)

$$\gamma ALS = LPKV^2 \qquad \Longrightarrow \quad V = \left(\frac{\gamma}{K}\right)^{\frac{1}{2}} \sqrt{RS} \tag{2.4}$$

Let $\sqrt{\gamma/_K}$ be replaced by a factor C; then equation (2.4) is reduced to the Chezy formula, or

$$V = C\sqrt{RS} \tag{2.5}$$

Here C is the coefficient which represents the resistance of the flow (Chezy coefficient). This coefficient is dependent on factors such as "V" average velocity, "R" hydraulic radius, channel roughness and viscosity. It is not dimensionless but has dimensions of acceleration, i.e. length/time^2 or LT^{-2}.

2.1.2 The Manning formula

The formula that is used most in open channels is Manning formula. This formula was developed empirically, Bilgil (1998). Through the studies he performed, Manning (1895) searched for a dimensionless velocity equation and proposed following formula.

$$V = CR^{\frac{2}{3}}S^{\frac{1}{2}} \tag{2.6}$$

This was further modified by others and expressed in metric (SI system) units as

$$V = \frac{1}{n}R^{\frac{2}{3}}S^{\frac{1}{2}} \tag{2.7}$$

As was the case with the Chezy resistance coefficient, n is not dimensionless but has dimensions of $TL^{-\frac{1}{3}}$ or in the specific case of the equation above $s/m^{\frac{1}{3}}$, where m is the dimension for length (metres).

Thus in general, Manning's Equation is expressed mathematically as:

$$V = \frac{\varphi}{n}R^{\frac{2}{3}}S^{\frac{1}{2}} \tag{2.8}$$

where φ is the conversion factor and $\varphi = 1$.

Manning's Equation above is an empirical equation in which the values of constants and exponents have been derived from experimental data of turbulent flow conditions. According to Manning's Equation, the mean velocity of flow is a function of the channel roughness, the hydraulic radius, and the slope of the energy gradient. As noted previously, for uniform flow, the slope of the energy gradient is assumed to be equal to the channel bottom slope.

Equating the Chezy's equation (2.5) and the Manning formula (2.8) yields the relation between the coefficient which represents the resistance of the flow (Chezy coefficient) and the Manning's coefficient i.e.

$$C = \frac{1}{n} R^{\frac{1}{6}} \tag{2.9}$$

Given the velocity, the discharge is calculated as the product of velocity and cross-sectional area

$$Q = AV \tag{2.10}$$

Substituting equation (2.7) in equation (2.10) yields

$$Q = KAR^{\frac{2}{3}} \tag{2.11}$$

where

$$K = \frac{1}{n} S^{\frac{1}{2}} \tag{2.12}$$

Equation (2.11) gives the flow rate through a channel of a given radius, slope and roughness, where K is a constant known as the roughness coefficient . Equation (2.11) indicates that the discharge Q is proportional to $AR^{\frac{2}{3}}$.

$$Q = KAR^{\frac{2}{3}} = KA\left(\frac{A}{P}\right)^{\frac{2}{3}} = K\frac{A^{\frac{5}{3}}}{P^{\frac{2}{3}}} \tag{2.13}$$

Substituting equations (1.1) and (1.3) in equation (2.13) and for a given constant K, the discharge Q through a circular channel can be written in terms of the angle between to radii θ and the radius r of the conduit as:

$$Q = K\frac{[r^2(\theta - \sin\theta\cos\theta)]^{\frac{5}{3}}}{(2r\theta)^{\frac{2}{3}}} = \frac{Kr^{\frac{10}{3}}\left(\theta - \frac{1}{2}\sin 2\theta\right)^{\frac{5}{3}}}{(2r\theta)^{\frac{2}{3}}} \tag{2.14}$$

2.3 GOVERNING EQUATIONS

Open-channel flows involve a free surface that can deform from its undisturbed relatively flat configuration to form waves. The basic equations that describe the propagation of a wave in an open channel are the Saint Venant' s equations. Those waves can be classified as: dynamic wave, gravitational wave, diffuse wave and cinematic wave according to the number of elements considered in the model.

These equations are the most commonly used system in the case of modeling predominantly one dimensional flows, and they describe the gradually varied flow of an incompressible inviscid fluid. These equations consist of continuity or mass equation, and an equation of motion derived from Newton's second law of motion along the channel.

2.4 ASSUMPTIONS

In this study, the following assumptions will be utilized.

i. The flow is one-dimensional such that the main component of velocity u, is along the x-axis and is a function of x alone.
ii. The fluid is Newtonian.
iii. Forces causing the flow are due to gravity.
iv. The water is considered as incompressible.
v. The flow is unsteady.

2.5 CONTINUITY EQUATION

A continuity equation is a differential equation that describes the transport of some kind of conserved quantity, in particular-mass. All examples of continuity equations express the same idea that the total amount (of the conserved quantity) inside any region can only change by the amount that passes in or out of the region through the boundary. A conserved quantity cannot increase or decrease, it can only move from place to place.

The continuity equation governing unsteady flow in open channels of arbitrary shape is:

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} = q \tag{2.15}$$

Substituting equation (2.10) in equation (2.15) above and then differentiating partially with respect to x yields:

$$V\frac{\partial A}{\partial x} + A\frac{\partial V}{\partial x} + \frac{\partial A}{\partial t} - q = 0 \tag{2.16}$$

The flow area is assumed to be a known function of the depth; therefore derivatives of A may be expressed in terms of y or :

$$\frac{\partial A}{\partial x} = \frac{dA}{dy}\frac{\partial y}{\partial x} = T\frac{\partial y}{\partial x} \tag{2.17}$$

$$\frac{\partial A}{\partial t} = \frac{dA}{dy}\frac{\partial y}{\partial t} = T\frac{\partial y}{\partial t} \tag{2.18}$$

In this discussion, it is assumed that T is determined by

$$T = \frac{dA}{dy} \qquad \text{Franz (1982)}$$

Substituting equations (2.17) and (2.18) in equation (2.16) yields:

$$VT\frac{\partial y}{\partial x} + A\frac{\partial V}{\partial x} + T\frac{\partial y}{\partial t} - q = 0 \tag{2.19}$$

or

$$\frac{\partial y}{\partial t} + \frac{A}{T}\frac{\partial V}{\partial x} + V\frac{\partial y}{\partial x} - \frac{q}{T} = 0 \tag{2.20}$$

2.6 MOMENTUM EQUATION

Momentum equations describe the motion of fluid substances. These equations are derived from the Newton’s second law of motion, together with the assumption that fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term. These equations relate the sum of forces acting on an element of fluid to its acceleration or rate of change of momentum.

These equations are non-linear partial differential equations. The non-linearity is due to convective acceleration, which is an acceleration associated with the change in velocity over position. Together with supplemental equations (for example, conservation of mass) and well formulated boundary conditions, the momentum equation model fluid motion accurately.

The momentum equation governing unsteady flow in open channels of arbitrary shape is:

$$\frac{\partial V}{\partial t} + \alpha V \frac{\partial V}{\partial x} + g \frac{\partial y}{\partial x} = g\left(S_o - S_f\right) \tag{2.21}$$

The channel bottom slope, S_o can be conveniently expressed as:

$$S_o = -\frac{dz}{dx} \tag{2.22}$$

where z is bed level or channel bottom elevation relative to a datum . The term dz/dx is the change of elevation of the bottom of the channel with respect to distance or the bottom slope.

The friction slope, S_f also known as the friction term due to bed's roughness is expressed as:

$$S_f = -\frac{dH}{dx} \tag{2.23}$$

where H is the total energy at any cross section of the channel. The term dH/dx is the change of energy with longitudinal distance or the friction slope.

Rearranging equation (2.21) yields:

$$\frac{\partial V}{\partial t} + \alpha V \frac{\partial V}{\partial x} + g \frac{\partial y}{\partial x} - g(S_o - S_f) = 0 \tag{2.24}$$

Equations (2.20) and (2.24) are non-linear first order partial differential equation of hyperbolic type.

2.7 METHOD OF SOLUTION

Equations (2.20) and (2.24) are non-linear first order partial differential equations of hyperbolic type. It is not possible to solve these equations analytically. The finite difference method will be used to obtain approximate solutions. In this technique, the partial derivatives in the equations are replaced by their corresponding finite difference approximations which results in a series of algebraic equations that can be solved on a computer. The solution to the discrete problem represents an approximation to the solution of the continuous problem.

Many techniques are available for numerical simulation work and in order to quantify how well a particular numerical technique performs in generating a solution to a problem, there are four fundamental criteria that can be applied to compare and contrast different methods. The concepts are accuracy, consistency, stability and convergence.

Accuracy is a measure of how well the discrete solution represents the exact solution of the problem. Two quantities exist to measure this – the local or truncation error, which measures how well the difference equations match the differential equations, and the global error which reflects the overall error in the solution. This is not possible to find unless the exact solution is known.

A technique is consistent if the truncation error decreases as the step size is reduced, that is to say that as $\Delta t, \Delta x$ tend to zero then the discretised equations should tend towards the differential equations. A technique is stable if any errors in the solution will remain bounded. In practice if an unstable method is used then the solution will tend towards infinity. Another requirement is that the numerical technique should be convergent, which means that the numerical solution should approach the exact solution as the grid spacing is reduced to zero.

2.8 FINITE DIFFERENCE METHOD

The finite difference approximations of the partial derivatives appearing in equations (2.20) and (2.24) are obtained by performing Taylor series expansion of the dependent variable and substituting the truncated expressions into the differential equation. The differentials are approximated by differences in the solution at various points. By definition,

$$u_x = \left(\frac{\partial u}{\partial x}\right) = \lim_{\Delta x \to 0} \frac{u(x + \Delta x) - u(x)}{\Delta x} \tag{2.25}$$

When Δx is small, this formula can be used as an approximation for the derivative of u at x. From Taylor series

$$u(x + \Delta x) = u(x) + \Delta x\, u_x(x) + \frac{\Delta x^2}{2} u_{xx}(x) + \cdots \tag{2.26}$$

By rearrangement

$$\frac{u(x + \Delta x) - u(x)}{\Delta x} = u_x(x) + \frac{\Delta x}{2} u_{xx}(x) + \cdots \tag{2.27}$$

If Δx is small, the higher order terms in the expansion will decrease and so it is possible to write

$$u_x(x) = \frac{u(x + \Delta x) - u(x)}{\Delta x} + 0(\Delta x) \tag{2.28}$$

From equation (2.28), the leading term of the error in approximating u_x by the right hand side is of order Δx and so this represents a first order approximation. It is possible to define other difference formula to approximate derivatives and these may have different orders of accuracy.

The above analysis deals with the continuous solution however the objective is to calculate u at a set of discrete points on the mesh, and this is the numerical solution. The numerical solution of equations (2.20) and (2.24) will be approximated at a discrete number of points arranged to form a rectangular grid. This rectangular grid is obtained by dividing the (x, t) plane into a network of rectangles of sides Δx and Δt by drawing the set of lines:

$$\left.\begin{array}{ll} x = i\Delta x = ih, & i = 0,1,2,\dots \\ t = j\Delta t = jk, & j = 0,1,2,\dots \end{array}\right\} \tag{2.29}$$

The nodes or mesh points or grid points of the network occur at the intersections of straight lines drawn parallel to the x and t axes. The lines parallel to the t axis represent locations along the channel while those drawn parallel to the x axis represent times. The location lines are drawn with spacing Δx while the time lines are drawn with spacing Δt.

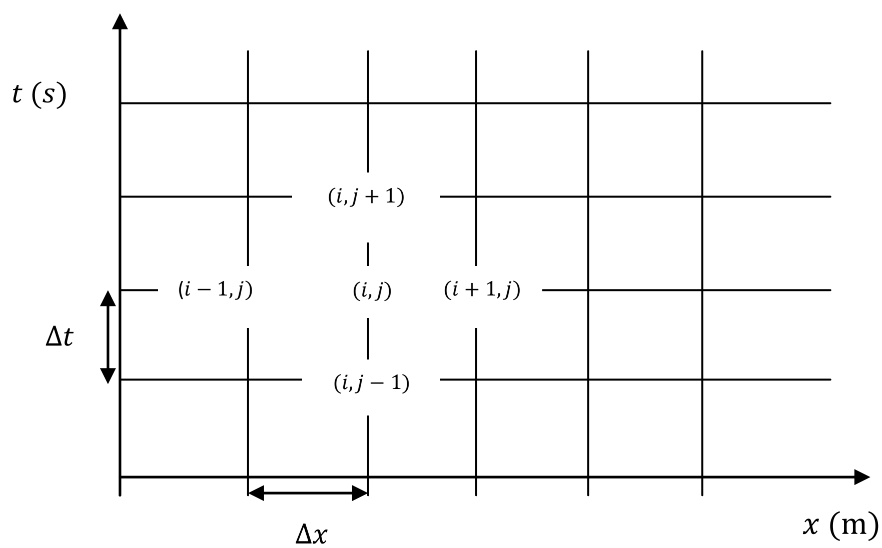


Figure 2.2: Finite difference mesh

Each node in the network is identified by two indices; the first designates spatial point (location) of the node in time while the second designates the time. Let $u_{i,j}$ be the numerical approximation to $u(x_i, t_j)$.

The forward difference for the first order derivatives of V with respect to time t is given by:

$$V_t = \frac{V_{i,j+1} - V_{i,j}}{k} + 0(k) \tag{2.30}$$

Whereas that of y with respect to time t is given by

$$y_t = \frac{y_{i,j+1} - y_{i,j}}{k} + 0(k^2) \tag{2.31}$$

Similarly the forward difference for the first order derivatives of V with respect to x is given by:

$$V_x = \frac{V_{i+1,j} - V_{i,j}}{h} + 0(h) \tag{2.32}$$

Whereas that of y with respect to x is given by:

$$y_x = \frac{y_{i+1,j} - y_{i,j}}{h} + 0(h) \tag{2.33}$$

The finite difference analogies of the partial differential equations can now be obtained by replacing the derivatives in the governing equations (2.20) and (2.24) by their corresponding difference approximations.

CHAPTER THREE

3.0 INTRODUCTION

In this chapter, the governing equations are presented in their finite difference forms. The effects of the various flow parameters on the flow velocity are presented graphically.

3.1 GOVERNING EQUATIONS IN FINITE DIFFERENCE FORM

Equations (2.20) and (2.24) are non-linear hence cannot be solved analytically. Therefore the finite difference method is used in their solution subject to the initial conditions

$$V(x,0) = V_0\,, \quad y(x,0) = y_0 \quad \text{for all } x > 0 \tag{3.1}$$

and boundary conditions

$$V(0,t) = V_0\,, \quad y(0,t) = y_0 \ \text{ for all } t > 0 \tag{3.2}$$

$$V(x_l,t) = V_0\,, \quad y(x_l,t) = y_0 \quad \text{for all } t > 0 \tag{3.3}$$

The primary difficulty with implicit finite difference techniques is the problem of numerically unstable solutions. Viessman *et al* (1972) noted that more stable solutions can be obtained if a diffusing difference approximation is used; i.e., in equations (2.20) and (2.24) substitute:

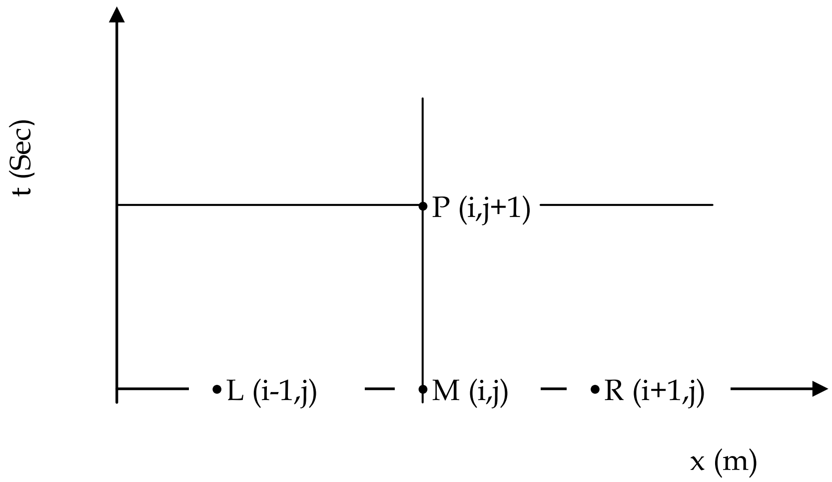


Figure 3.1: Mesh points

$$\frac{\partial V}{\partial t} = \frac{V(i,j+1) - 0.5\big(V(i-1,j) + V(i+1,j)\big)}{\Delta t} \tag{3.4}$$

$$\frac{\partial y}{\partial t} = \frac{y(i,j+1) - 0.5\big(y(i-1,j) + y(i+1,j)\big)}{\Delta t} \tag{3.5}$$

$$S_f = \frac{S_f(i-1,j) + S_f(i+1,j)}{2} \tag{3.6}$$

$$\frac{\partial V}{\partial x} = \frac{V(i+1,j) - V(i-1,j)}{2\Delta x} \tag{3.7}$$

$$\frac{\partial y}{\partial x} = \frac{y(i+1,j) - y(i-1,j)}{2\Delta x} \tag{3.8}$$

Substituting (3.5), (3.7) and (3.8) in equation (2.21) yields:

$$\frac{y(i,j+1)-0.5\big(y(i-1,j)+y(i+1,j)\big)}{\Delta t}+\frac{A}{T}\frac{V(i+1,j)-V(i-1,j)}{2\Delta x}$$

$$+V(i,j)\frac{y(i+1,j)-y(i-1,j)}{2\Delta x}-\frac{q}{T}$$

$$=0 \qquad (3.9)$$

$$y(i,j+1)=0.5[y(i-1,j)+y(i+1,j)]$$

$$-\Delta t\left\{\frac{A}{T}\frac{V(i+1,j)-V(i-1,j)}{2\Delta x}+V(i,j)\frac{y(i+1,j)-y(i-1,j)}{2\Delta x}\right.$$

$$\left.-\frac{q}{T}\right\} \qquad (3.10)$$

which is the finite difference form of the continuity equation.

Substituting equations (3.4), (3.6), (3.7) and (3.8) in equation (2.25)

$$\frac{V(i,j+1)-0.5\big(V(i-1,j)+V(i+1,j)\big)}{\Delta t}+\alpha V(i,j)\frac{V(i+1,j)-V(i-1,j)}{2\Delta x}$$

$$+g\frac{y(i+1,j)-y(i-1,j)}{2\Delta x}-g\left(S_0-\frac{S_f(i-1,j)+S_f(i+1,j)}{2}\right)$$

$$=0 \qquad (3.11)$$

In the computation of unsteady flow, it is usually assumed that the friction slope S_f can be estimated from either the Manning or Chezy resistance equations. In this work, we utilize the Manning resistance equation which is expressed as:

$$S_f=\frac{n^2V^2}{R^{4/3}} \qquad (3.12)$$

Substituting (3.12) in equation (3.11) yields:

$$
\begin{aligned}
V(i,j+1) = 0.5[V(i-1,j) + V(i+1,j)] \\
- \Delta t \left\{ \alpha V(i,j) \frac{V(i+1,j) - V(i-1,j)}{2\Delta x} + g \frac{y(i+1,j) - y(i-1,j)}{2\Delta x} \right. \\
- g \left[S_0 \right. \\
- \frac{n^2}{2R^{4/3}} \left(V^2(i-1,j) \right. \\
\left. \left. \left. + V^2(i+1,j) \right) \right] \right\} \qquad (3.13)
\end{aligned}
$$

In equations (3.10) and (3.13), the index i refers to spatial points whereas the index j refers to time. The consecutive terms of depth and velocities $y_{i,j+1}$ and $V_{i,j+1}$ respectively are computed by equations (3.10) and (3.13) subject to the initial condition

$$V(x,0) = 10\,, \quad y(x,0) = 0.5 \quad \text{for all } x > 0 \qquad (3.14)$$

and boundary conditions

$$V(0,t) = 10\,, \quad y(0,t) = 0.5 \quad \text{for all } t > 0 \qquad (3.15)$$

$$V(x_l,t) = 10\,, \quad y(x_l,t) = 0.5 \quad \text{for all } t > 0 \qquad (3.16)$$

The computations are performed using small values of Δt. In our research, we set $\Delta t = 0.0012 \; and \; \Delta x = 0.1$. From equation (3.10), the depth y at the end of the time step $\Delta t, y_{i,j+1}$ $i = 1,2,3, \ldots, 40$ is computed in terms of velocities and depths at points on earlier time step. Similarly, $V_{i,j+1}$ is also to be computed from equation (3.13). The procedure is to fix i, starting with $i = 1$ while varying j values from $j = 0$ till $j = 49$. This is repeated with $i = 2, 3, \ldots, 40$ with j values varying from $j = 0$ to $j = 49$ for each i. Using equations (3.10) and (3.13), the following results are obtained.

3.2 RESULTS

Equations (3.10) and (3.13) were solved using the computer code in Appendix I, by varying i and j as discussed in the previous section. The values of V against those of y were plotted giving the curve in figure 3.2 below. Moreover, by varying the specified parameters, the curves in figures 3.3 – 3.5 were obtained.

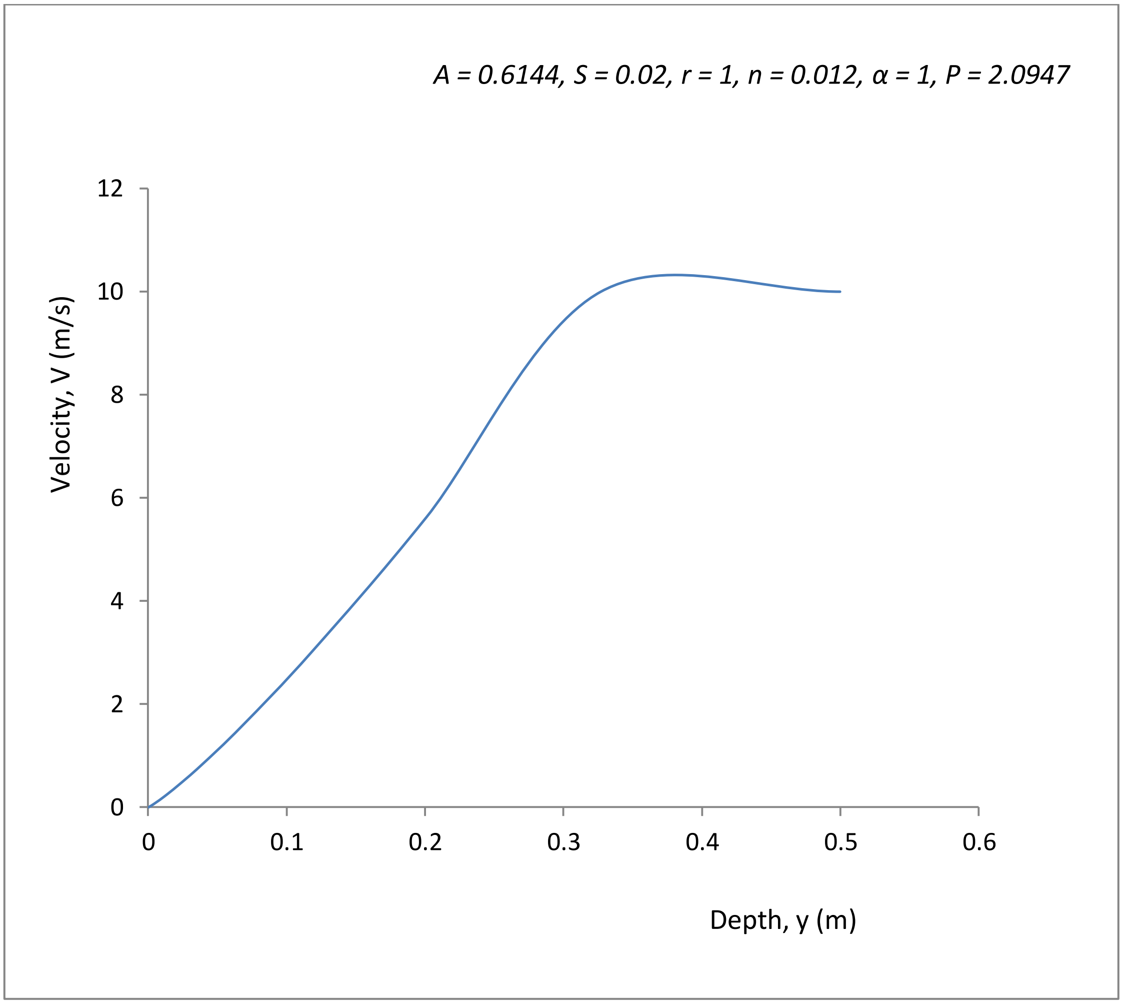


Figure 3.2: Velocity profiles versus depth.

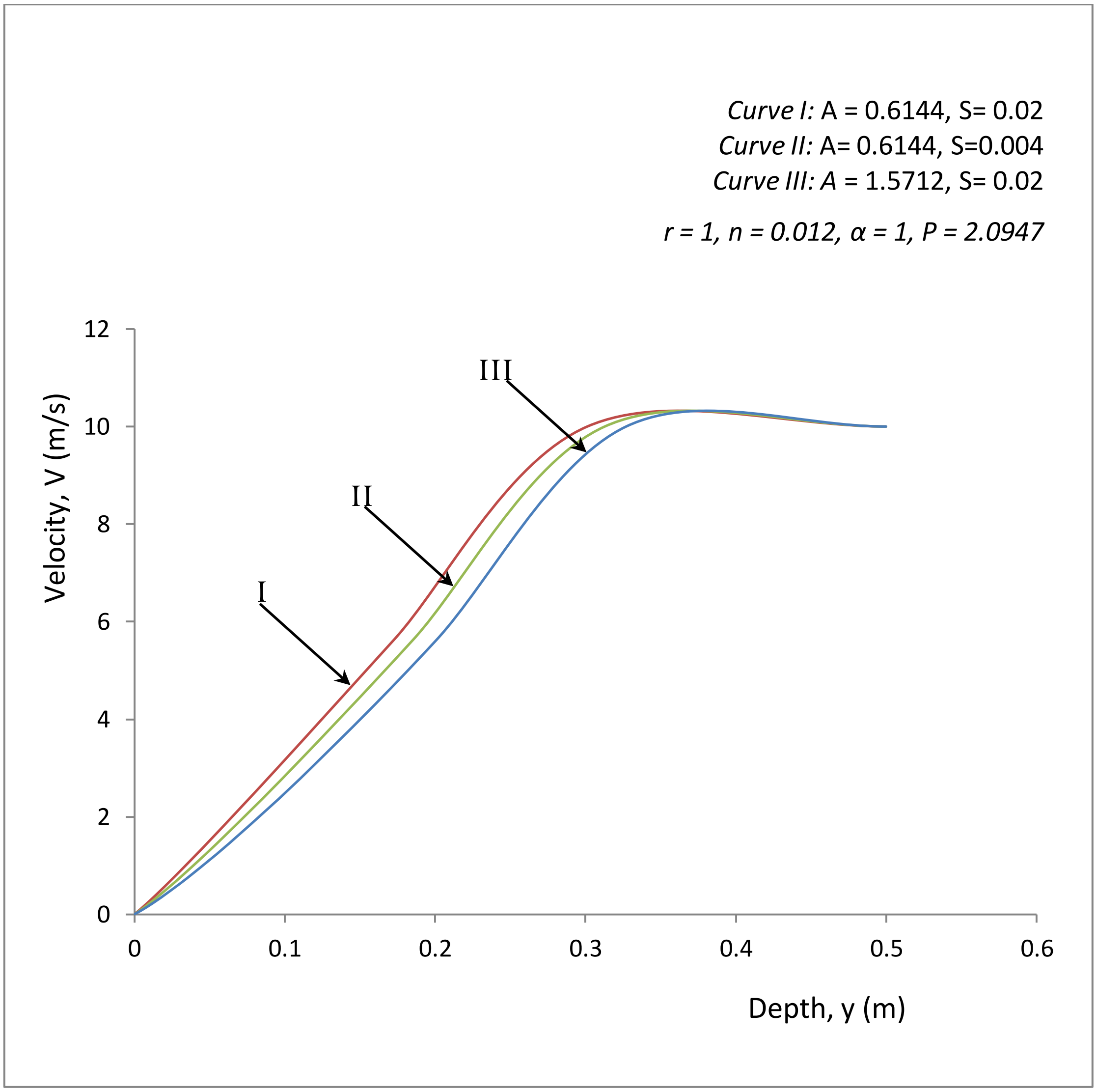


Figure 3.3: Velocity profiles versus depth for varying A and S.

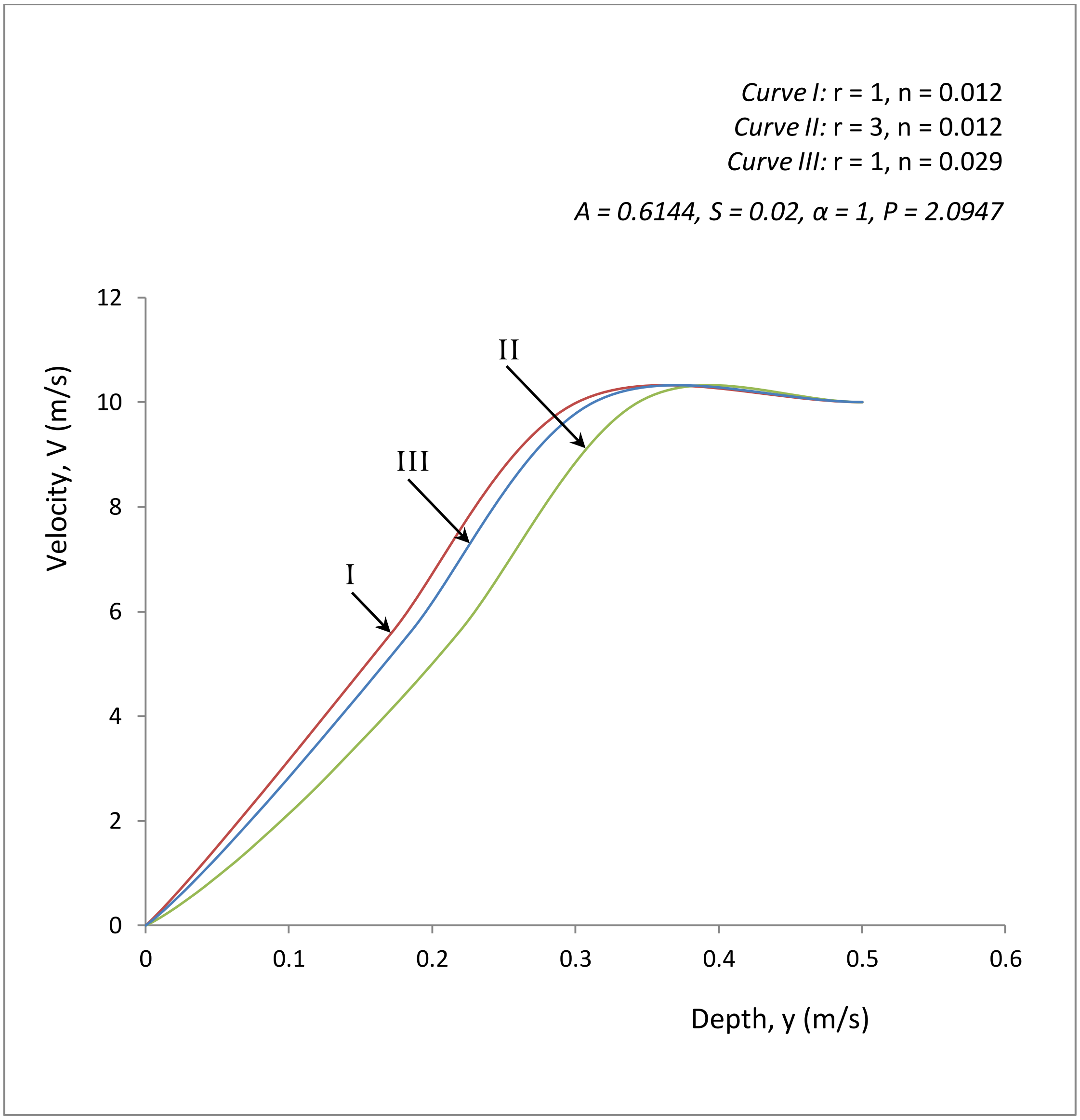


Figure 3.4: Velocity profiles versus depth for varying r and n.

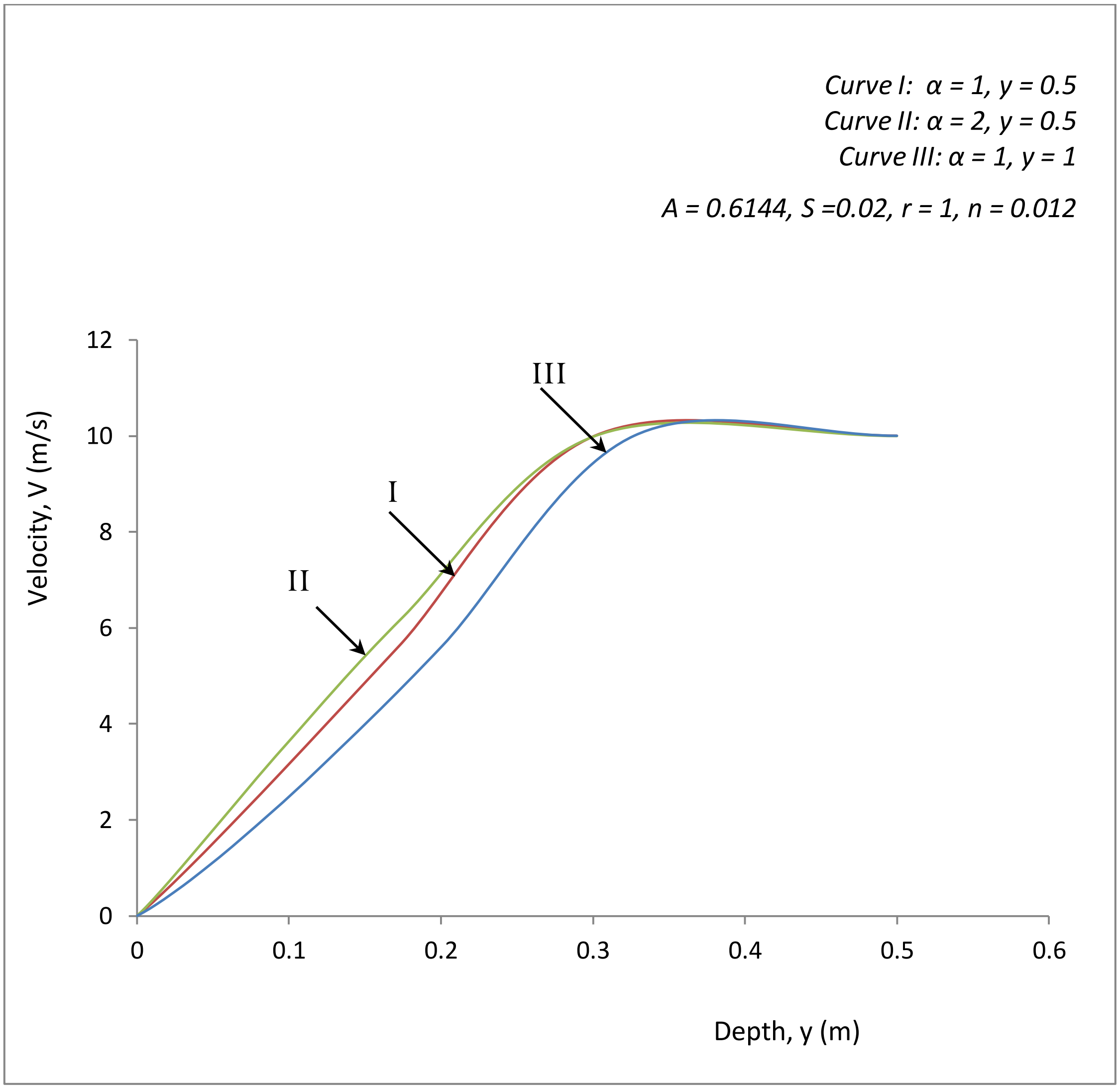


Figure 3.5: Velocity profiles versus depth for varying α and y.

3.3 DISCUSSION

From figure 3.2, we observe that velocity increases with increase in depth. The free surface occurs at a depth of 0.5m and the velocity of the fluid layer at this depth is 10m/s. It is also observed that maximum velocity occurs just below the free surface, at a depth of about 0.38m.

The flow velocity in a channel section varies from one point to another due to shear stress at the bottom and at the sides of the channel. The velocity is not maximum at the free surface mainly due to surface tension caused by the strong cohesive forces among the liquid molecules. In the bulk of the liquid, each molecule is pulled equally in every direction by neighbouring liquid molecules, resulting in a net force of zero. The molecules at the surface do not have other molecules surrounding them entirely and are therefore pulled inwards. This creates some internal pressure and forces liquid surfaces to contract to the minima leading to a reduction in velocity at the free stream. Second, both the atmospheric pressure and gravity creates some internal pressure causing the contraction of the liquid surface. This contraction lowers the movement of the fluid particles at the free surface resulting to reduced velocities. In addition, the wind blowing over the free surface also affects the velocity in the free stream due to frictional resistance particularly when wind blows over the free surface at high velocities and in the opposite direction to the main direction of flow.

From figure 3.3, we observe that a reduction in the slope from 0.02 m/m to 0.004 m/m leads to a decrease in the flow velocity as shown from curve I to curve II. An increase in the cross sectional area of flow from $0.6144m^2$ to $1.5712m^2$ results to a decrease in the flow velocity from curve I to curve III.

The velocity formula equation (2.7) shows a direct relationship between flow velocity and the slope. Thus a decrease in slope results to a decrease in the flow velocity. An increase in the cross-sectional area of flow leads to an increase in the wetted perimeter. A large wetted perimeter results to high shear stresses at the sides of the channel which results to a reduction in the flow velocity.

From figure 3.4, we observe that increasing the radius from 1m to 3m results to a decrease in the flow velocity from curve I to curve II. Moreover, an increase in the roughness coefficient from 0.012 to 0.029 also results to a reduction in the flow velocity as shown from curve I to curve III.

An increase in the radius results in an increase in the wetted perimeter because the fluid will spread more in the conduit. A large wetted perimeter will result to large shear stresses at the sides of the channel and therefore the flow velocity will be reduced. An increase in the roughness coefficient results to large shear stresses at the sides of the channel. This means that the motion of fluid particles at or near the surface of the conduit will be reduced. The velocity of the neighbouring molecules will also be lowered due to constant bombardment with the slow moving molecules leading to an overall reduction in the flow velocity.

From figure 3.5, we observe that an increase in the energy coefficient from 1 to 2 leads to an increase in the flow velocity from curve I to curve II. In addition, an increase in the flow depth from 0.5m to 1m results to a reduction in the flow velocity from curve I to curve III.

An increase in the energy of the fluid results in the molecules attaining high energy which leads to more random motion. This random motion causes constant bombardment between the fluid particles resulting to an increase in velocities of the molecules and in general, of the fluid. The increase in flow depth leads to an increase in the wetted perimeter. This results in large shear stresses at the sides of the channel and therefore the flow velocity will be reduced.

CHAPTER FOUR

4.0 INTRODUCTION

In this chapter, the description of the procedure followed in obtaining solutions to the governing equations is reviewed. Conclusions from the results obtained is also presented. The chapter closes with recommendations for further work.

4.1 CONCLUSIONS AND RECOMMENDATIONS

The objective of this thesis was to investigate the effects of the various flow parameters on the flow velocity. An analysis of the effects of the various parameters on the flow velocity has been carried out. The equations governing the flow considered in the problem are non-linear and therefore to obtain their solutions, an efficient finite difference scheme has been developed as outlined in chapter two. The mesh used in the problem considered in this work is divided uniformly.

The various flow parameters were varied, one at a time while holding the other parameters constant. This was repeated for all the flow parameters and the results presented graphically. It was established that for a fixed flow area, the flow velocity increases with increase in depth from the bottom of the channel to the free stream and that maximum velocity occurs just below the free surface. The flow velocity in a channel section varies from one point to another due to shear stress at the bottom and at the sides of the channel. The velocity is not maximum at the free surface mainly due to surface tension caused by the strong cohesive forces between the liquid molecules. In the bulk of the liquid, each molecule is pulled equally in every direction by neighboring liquid molecules, resulting in a net force of zero. The molecules at the free surface do not have other molecules surrounding them entirely and are therefore pulled inwards. This creates some internal pressure and forces liquid surfaces to contract to the minima leading to a reduction in velocity at the free stream Second, the both the atmospheric pressure and gravity acting in a direction which is perpendicular to the free surface creates some internal pressure causing the contraction of the liquid surface. This contraction lowers the movement of the fluid particles at the free surface resulting to reduced velocities. In addition, the wind blowing over the free surface also affects the velocity in the free stream due to frictional resistance particularly when wind blows over the free surface at high velocities and in the opposite direction to the main direction of flow.

Reduction in the slope leads to a decrease in the flow velocity since the slope and flow velocity are directly proportional. This is true as reflected by both the Chezy and Manning formulae discussed in chapter two. An increase in the cross sectional area of flow results to a decrease in the flow velocity. An increase in the cross-sectional area of flow leads to an increase in the wetted perimeter. A large wetted perimeter results to high shear stresses at the sides of the channel which results to a reduction in the flow velocity.

An increase in the radius of the conduit results to a reduction of the flow velocity. This is because, as the radius is increased, so is the wetted perimeter as the fluid spreads more in the conduit. A large wetted perimeter will result to large shear stresses at the sides of the channel and therefore the flow velocity will be reduced. Moreover, an increase in the roughness coefficient also results to a decrease in the flow velocity due to large shear stresses at the sides of the channel. This means that the motion of fluid particles at or near the surface of the conduit will be reduced. The velocity of the neighbouring molecules will also be reduced due to constant bombardment with the slow moving molecules leading to an overall reduction in the flow velocity.

An increase in the energy coefficient leads to an increase in the flow velocity due to an increase in the energy of the fluid resulting in an increase in the molecular energy which leads to more random motion. This random motion causes constant bombardment between the fluid particles resulting to increased velocities of the molecules and in general, of the fluid. Finally, an increase in the flow depth results to a reduction in the flow velocity. The increase in flow depth leads to an increase in the wetted perimeter resulting in large shear stresses at the sides of the channel with an effect of reduced velocities.

The results obtained in this work agrees with the results obtained experimentally which is presented in Appendix II.

Further work could include:

- Extension of the project to include irregular geometries.
- Implicit finite difference techniques may also be employed to solve the non-linear Saint Venant equations.
- Future work on the topic could address the extension of the model to natural channels with growing vegetation.
- Finally, more experimental tests are also recommended.

4.2 REFERENCES

Akbari G. and Firoozi B. (2010). *Implicit and Explicit Numerical Solution of Saint-Venant Equations for Simulating Flood Wave in Natural Rivers.* 5th National Congress on Civil Engineering. Ferdowsi University of Mashhad, Mashhad, Iran.

Bilgil A. (1998). *The effect of wall shear stress on friction factor in smooth open channel flows.* Ph. D. Thesis, Karadeniz Technical University, Trabzon, Turkey.

Çeçen K. (1982). *Hydraulic*: **2**. İstanbul Teknik Üniversitesi (*I.T.U).*

Chadwick A. and Morfet J. (1993). *Hydraulics in Civil Engineering and Environmental Engineering*. Chapman & Hall: 187 – 200.

Chagas P. and Souza R. (2005). *Solution of Saint Venant's Equation to Study Flood in Rivers, through Numerical Methods.* Paper_1 Conf AGU Hydrology Days.

Chamkha A. J. (2010). *Heat and Mass Transfer of a Non-Newtonian Fluid Flow over a Permeable Wedge in Porous Media with Variable Wall Temperature and Concentration and Heat Source or Sink. WSEAS Transactions On Heat And Mass Transfer* **5**: 11 – 20.

Chow V. T. (1959). *Open-Channel Hydraulics.* McGraw Hill Book Company. New York. 1 – 40.

Crossley A. J. (1999). *Accurate and efficient numerical solutions for the Saint Venant equations of open channel flow*. Ph.D thesis. University of Nottingham, United Kingdom.

Franz D. D. (1982). *Tabular representation of cross-sectional elements. Proceedings of the American Society of civil engineers*. *Journal of the hydraulics division.* **108:** 1070 – 1081.

French, R. H. (1985). *Open-Channel Hydraulics.* Mc-Graw Hill, Inc. New York.

Henderson F. M. (1966). *Open Channel Flow*. Macmillan Publishing Company. New York.

Khan A. A. (2000). *Modeling flow over an initially dry bed*. *Journal of Hydraulic Research* **38**: 383 – 388.

Kwanza J. K., Kinyanjui M. and Nkoroi J. M. (2007). *Modeling fluid flow in rectangular and trapezoidal open channels. Advances and Applications in Fluid Mechanics.* **2**: 149 – 158.

Manning R. (1895). *Flow of Water in Open Channels Supplement*. *Institution of Civil Engineers of Ireland,* **24**: 179 – 207.

Moshirvaziri S., Sheridan G., Lane P. and Jones O. *Modelling connectivity between pollutant source areas and streams.* http://www.iemss.org/iemss2010/index.php?n=Main.Proceedings. November 2010.

Nalluri C. and Adepoju B. A. (1985). Shape effects on resistance to flow in smooth channels of circular cross-section. *Journal of Hydraulic Research* **23**: 37 – 46.

Rott N. (1990). *Note on the history of the Reynolds number. Annual Review of Fluid Mechanics.* **22**: 1 – 11.

Sinha P. C. and Meena A. (1982). *Entry flow in a straight circular pipe. Journal of Australian Mathematics* (Series B) **24:** 59 – 66.

Stokes G. (1851). *The effect of the Internal Friction of Fluids on the Motion of Pendulums. Transactions of the Cambridge Philosophical Society* **9**: 8 – 106.

Swayne D. A., Yang W., Voinov A. A., Rizzoli A. and Filatova T. (2010). *International Congress on Environmental Modelling and Software Modelling for Environment's Sake.* Fifth Biennial Meeting, Ottawa, Canada.

Tuitoek D. K. and Hicks F. E. (2001). *Modelling of unsteady flow in compound channels. African Journal Civil Engineering* **4**, pp. 45-53.

Viessman W., Jr., Knapp J. W., Lewis G. L. and Harbaugh T. E. (1972). *Introduction to hydrology, Second edition*. Harper & Row. New York, pp. 1 – 60.

Appendix I: COMPUTER CODE

In order to solve the governing equations (3.10) and (3.15), the following computer program code was developed using Visual Basic subject to the the initial condition (3.14) and boundary conditions (3.15) and (3.16). The results were obtained by varying various flow parameters, notably cross-sectional area A of flow, channel radius r, roughness coefficient n, energy coefficient α and slope S of the channel.

```
Private Sub Command1_Click()Dim
 Y(0 To 40, 0 To 50), V(0 To 40, 0 To 50), q As Single, b As Single, g As Single
Dim delX As Double, delT As Double, I As Integer, J As Integer, ITMAX As Single, N As Integer
Dim ITCOUNT As Integer, theta As Single, So As Single
Dim FILENUM As Byte, T As Single, P As Single, r As Single, A As Single, alpha As Single, beta As
Single, Angle As Single, ni As Single
N = 40: delX = 0.1: q = 0: g = 9.81: So = 0.02:  r = 1: ni = 0.012: alpha = 1
'N is the no. of subdivisions along the channel.
delT = 0.0012
'*******************************************************
ITMAX = 50   'no of subdivisions on the time
FILENUM = FreeFile()
Angle = CSng(InputBox("Peter, Enter the Angle in degs", "ANGLE!", 0))
theta = Angle * 3.142 / 180
A = (theta - Sin(theta) * Cos(theta)) * r ^ 2
T = 2 * r * Sin(theta)
P = 2 * r * (theta)
Open "C:\Documents and Settings\da\Desktop\PETER_4\PeterProfiles.txt" For Append As FILENUM
```

```
Rem Initial condition

For I = 0 To N

For J = 0 To ITMAX      'J is time

        V(I, 0) = 10: Y(I, 0) = 0.5

Next

Next

 '  Boundary conditions

For I = 0 To N

For J = 0 To ITMAX

  V(0, J) = 10: Y(0, J) = 0.5      'entry values

    V(40, J) = 10: Y(40, J) = 0.5     'exit values

Next

Next

'Solving for velocities

For I = 1 To N - 1

  For J = 0 To ITMAX - 1

    'calculate Y

    Y(I, J + 1) = 0.5 * (Y(I - 1, J) + Y(I + 1, J)) - delT * ((A / T) * (V(I + 1, J) - V(I - 1, J)) / (2 * delX) +
V(I, J) * (Y(I + 1, J) - Y(I - 1, J)) / (2 * delX) - q / T)

    'calculate U

    V(I, J + 1) = 0.5 * (V(I - 1, J) + V(I + 1, J)) - delT * (alpha * V(I, J) * (V(I + 1, J) - V(I - 1, J)) / (2 *
delX) + g * (Y(I + 1, J) - Y(I - 1, J)) / (2 * delX) - g * (So - ((0.5 * ni ^ 2 / (A / P) ^ (4 / 3)) * (V(I - 1, J) ^
2 + V(I + 1, J) ^ 2))))

Next

Next

Print #FILENUM, I, "q = " & q
```

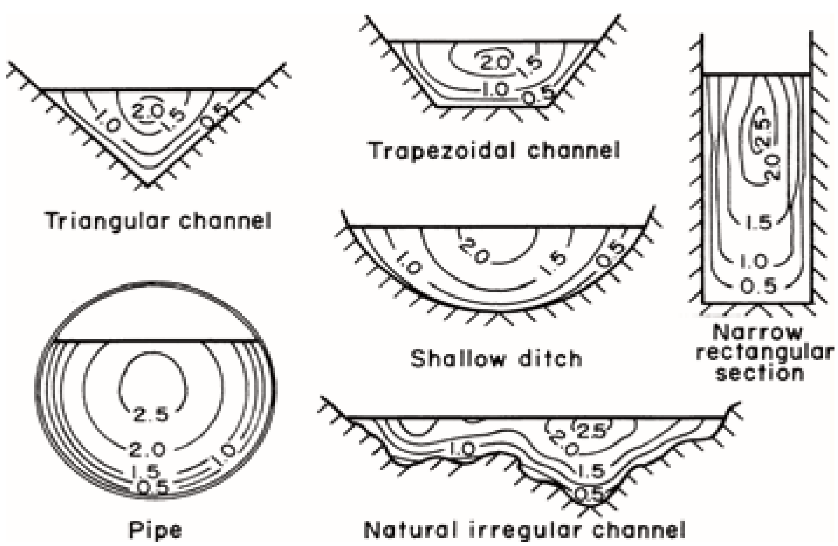
```
For J = 0 To ITMAX
For I = 0 To N
   Print #FILENUM, V(I, J);
      If I = N Then Print #FILENUM, vbCrLf;
Next
Next
'************************unsteady values
Close #FILENUM
MsgBox "AM THROUGH!!!", vbCritical, "Peter"
End Sub
Private Sub Command2_Click()
On Error GoTo kan
Kill "C:\Documents and Settings\da\Desktop\PETER_4\PeterProfiles.txt"
kan:
Exit Sub
End Sub
```

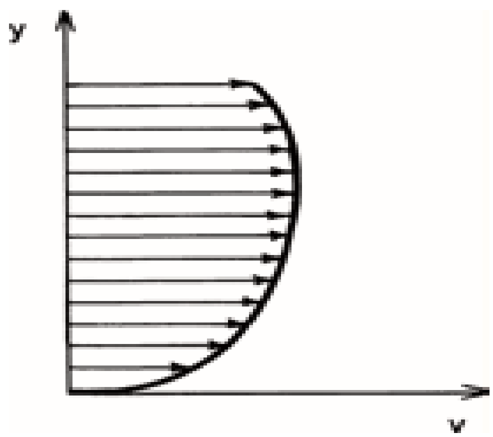
Appendix II: VALIDATION

The flow velocity in a channel section varies from one point to another. This is due to shear stress at the bottom and at the sides of the channel and due to the presence of free surface. The figures below show typical velocity distributions in different channel cross sections obtained experimentally. The flow velocity may have components in all three Cartesian coordinate directions. This velocity component varies with depth from the free surface.

The results obtained in this work agrees with the results given below which were obtained experimentally and which gives velocity distribution in different channel sections:

(After Chow [1959])




y
v