

The copyright of this thesis vests in the author. No quotation from it or information derived from it is to be published without full acknowledgement of the source. The thesis is to be used for private study or non-commercial research purposes only.

Published by the University of Cape Town (UCT) in terms of the non-exclusive license granted to UCT by the author.

17

**-A computational fluid dynamics model for sediment
movement based on the unit stream power approach**

by

Caroline M^c Gahey

**A thesis submitted in partial fulfillment of the requirements for
the degree of Master of Science in Engineering**

Department of Civil Engineering

University of Cape Town

August 2001

Declaration

I, Caroline M^c Gahey, hereby declare that unless otherwise indicated the work in this thesis is my original work and that neither the whole work nor any part of it has been, is being or is to be submitted for another degree in this or any other university.

I empower the university to reproduce for the purpose of research either the whole or any portion of the contents in any manner whatsoever.

Signed by candidate

SIGNATURE

31 August 2001

DATE

Acknowledgements

I would like to thank the following people for their support throughout the dissertation period:

- (i) Neil Armitage for his continued support and guidance.
- (ii) Howard Pearce for his invaluable guidance and direction throughout.
- (iii) David Fletcher for his excellent assistance with CFX.
- (iv) John Roux for his continued support with CFX.
- (v) Kourosh Babaeyan-Koopaei for providing detailed sediment and flow data.
- (vi) Kazuhiro Fujisaki for providing detailed experimental data.
- (vii) Massimo Cellino for providing experimental data of shear stresses over dunes.
- (viii) The Water Research Commission of South Africa for providing financial assistance.
- (ix) And most importantly, my mom, for providing continued support and encouragement throughout this thesis period.

Terms of Reference

Neil Armitage of the Civil Engineering Department at the University of Cape Town commissioned this thesis in October 1999. This study forms an integral part of a report to the Water Research Commission of South Africa, which has provided funding for the study of sediment transport around hydraulic structures in rapidly varied flow.

The study brief for this thesis includes the prediction of scour around engineering structures in open channel flow using a three-dimensional numerical model based on the unit stream power approach. The numerical model is solved by means of a commercially available software, CFX-V4.3 (1999) and CFX-V4.4 (2001). These scour predictions are validated against small-scale laboratory experiments.

Abstract

Scour profiles around engineering structures in open channel flow are an essential parameter in foundation design. Excessive scour around structures such as bridge piers can progressively undermine the foundations resulting in structural failure and ultimately, collapse. Complete protection against scour is expensive and therefore not generally a feasible design option. Consequently the maximum scour depth and upstream scour profiles should be predicted to minimise the risk of failure. Scour prediction methods are predominantly empirically based due to the complex physics of the scour processes. Alternatively, small-scale hydraulic models are widely used.

This thesis introduces a three-dimensional numerical model based on the unit stream power approach for the prediction of scour profiles. The unit stream power approach is employed to define the incipient motion criteria for the sediment particles by comparing the available applied stream power with the individual particle power requirements for motion (Rooseboom 1992). The Movability Number (Lui 1957), which represents the shear velocity – settling velocity ratio, is related to the unit stream power and is applied as an alternative incipient motion criterion. The Movability Number, as adapted by Armitage (2001) to account for topography, is utilised for scour hole predictions.

The presence of hydraulic structures in an alluvial channel alters the natural flow pattern causing channel constriction and steep velocity gradients, which increase the applied stream power and thus the erosive potential of the water. The numerical model is an important tool for determining the flow characteristics adjacent to these obstructions and hence the available stream power. The commercial flow solver CFX-V4.3 (1999), which is based on the finite volume method, is adapted to simulate open channel flow conditions and generate contours of the predicted scour. These scour profiles are validated by comparison to observed scour patterns in small-scale laboratory models. CFX-V4.4 (2001), which was released in the concluding stages of this research, was implemented for the final, most complex application.

In the numerical model, the free surface was modeled with a moving wall boundary condition. The QUICK differencing scheme was implemented for the discretisation of the velocity and pressure terms, while the HYBRID scheme was employed for solution of the turbulent quantities. The standard $k-\varepsilon$ and the low Reynolds number $k-\varepsilon$ models were found to predict the turbulent shear stresses and viscosities with reasonable accuracy for both two- and three-dimensional open channel flow-fields. The analyses were conducted using steady state simulation. The resulting numerical model was then used to predict the onset or cessation of particle motion on the channel bed and hence the likelihood of local scour or deposition respectively.

The numerical model was tested against the movement of beads and the scour profiles of sand particles around hydraulic structures as observed in laboratory experiments. The movement of beads over a vertical weir, under an underflow structure and through a sudden contraction and expansion were numerically predicted. These were compared to the experimental observations (Offerman 1998; Feuilloley 1999; Lyons 1999). The numerical model was found to under-predict the power required for suspension at certain points along the flow path, however, once suspended the beads followed the streamlines and traversed the path as recorded in the physical data.

Scour predictions were numerically generated for scour profiles above a vertical weir, around a vertical-wall abutment and a cylindrical pier in a rigid, rectangular channel and around a cylindrical pier in a self-formed channel. These were compared to observed equilibrium scour patterns in small-scale laboratory models (Babaeyan-Koopaei 1996; Harnett 1998; Mitchell 2000; Midgley 2000). The numerical model predicted the size and shape of the scour regions reasonably well, however, for the cylindrical pier in the rigid rectangular channel the scour region was slightly shifted towards the centre of the channel. This shift was attributed to the surface being modeled as a horizontal moving wall, which tends to raise the pressure on the line of symmetry above what it would be in reality. In reality a slight drop in surface water elevation and sediment deposition characterises the downstream region.

Although the critical applied unit stream power regions were not identical to those predicted by the critical Movability Number, the regions were similar in size and

shape. The scour hole predictions for the pier situated in the self-formed channel were modeled in the initial pre-scour state and in the final equilibrium state. The Movability Number that was adapted for topography was found to predict a strong scour potential for the initial pre-scour state and approximately zero scour potential in the scour hole region for the final equilibrium state.

The results of the numerical computations correlate reasonably well with the experimental data, thus showing the potential of the CFD model as an alternative to laboratory experiments in the prediction of local scour. However, full implementation of this approach requires automation of the grid generation to reduce the modeling time and facilitate the gradual erosion of the scour profile for different flow rates. A free-surface algorithm should also be implemented.

Table of Contents

Declaration	ii
Acknowledgements	iii
Terms of reference	iv
Abstract	v
List of figures	xiii
List of symbols	xvii
List of tables	xxi
1. Introduction	1
2. Open channel flow characteristics	6
2.1 Co-ordinate system	6
2.2 Density, shear stress and viscosity	6
2.3 Reynolds stress model	8
2.4 The Prandtl eddy model	11
2.5 Free, forced and Rankine vortices	12
2.6 The boundary layer	13
2.7 Effect of roughness on the boundary layer	17
2.8 Alternative velocity distribution for turbulent flow	20
2.9 Conclusions	21
3. Incipient motion criteria	22
3.1 The sediment particles	22
3.1.1 Cohesiveness	22
3.1.2 Particle size	22
3.1.3 Particle density	23
3.1.4 Particle settling velocity	23
3.1.5 Angle of repose	25

3.1.6	Application of the sediment properties	25
3.2	Sediment motion	26
3.2.1	The critical velocity approach	27
3.2.2	The shear stress approach	28
3.3	One-dimensional unit stream power approach	29
3.3.1	Minimum rate of energy dissipation	30
3.3.2	Stream power distribution in an infinitely wide channel	30
3.3.3	The Movability Number as an incipient motion criterion	35
3.3.4	Rigid body analogy to applied power in turbulent flow	38
3.4	Three-dimensional unit stream power distribution	39
3.4.1	The fluid as a continuum	39
3.4.2	Mass conservation	41
3.4.3	Substantive derivative	41
3.4.4	The momentum equation	42
3.4.5	The energy equation	45
	(a) Energy flux to a fluid particle	45
	(b) Internal and kinetic energy components	46
3.4.6	The dissipation function and stream power	48
3.5	The angle of repose and stream power	49
3.6	Conclusions	50
4.	Scour and deposition around hydraulic structures	51
4.1	Scour types	51
4.2	Vertical weirs	54
4.2.1	Flow patterns around a vertical weir	54
4.2.2	Prediction of particle movement and estimation of scour depths	55
4.2.3	Flow under an 'underflow structure'	56
4.3	Scour around cylindrical piers	57
4.3.1	Flow patterns around a cylindrical pier	57
4.3.2	Estimation of scour depth	58
4.3.3	Numerical models	60

4.4	Scour around vertical-wall abutments	61
4.4.1	Abutment types	61
4.4.2	Flow patterns around a vertical-wall abutment	61
4.4.3	Estimation of scour depth	63
4.4.4	Numerical models of predicted flow fields and scour	64
4.5	Conclusions	66
5.	Computational fluid dynamics	67
5.1	CFD coding	67
5.2	The governing equations of fluid flow	68
5.3	Initial and boundary conditions	69
5.4	Turbulence and the governing equations	70
5.4.1	Necessity for turbulence modelling	70
5.4.2	The Reynolds averaged Navier-Stokes equations	71
5.4.3	Available models in CFX-V4.3	73
5.4.4	Applicability of models in different flow situations	73
5.5	Differencing schemes	77
5.6	Pressure correction	79
5.7	Solution control	80
5.8	Transient flow	81
5.9	Conclusions	82
6.	Adaptation of CFX-V4.3 (1999) for open channel flows	83
6.1	Mesh generation	83
6.1.1	Creating the geometry	83
6.1.2	Refining the mesh at the bed	84
6.1.3	Refining the mesh in area of interest	85
6.2	Programming response	86
6.3	Boundary conditions	88
6.3.1	Inlet velocity profile	88

6.3.2	Outlet	91
6.3.3	Symmetry	92
6.3.4	Walls	92
6.3.5	Free surface	92
6.4	Shear stress distribution with depth	97
6.5	Differencing scheme selection	99
6.6	Turbulence criteria selection	102
6.6.1	Length scale, intensity and eddy iterations	102
6.6.2	Selection of turbulence model	104
6.7	Pressure correction	107
6.8	Solvers	108
6.9	Buoyancy	109
6.10	Steady and transient flow	109
6.11	Topography adaptation	111
6.12	Final methodology	115
7.	Numerical predictions of the bead movement	119
7.1	Movement over a vertical weir	119
7.2	Movement under an underflow structure	123
7.3	Movement through a contraction and over a weir	129
7.4	Movement through an expansion and over a weir	136
7.5	Conclusions	140
8.	Numerical predictions of scour and deposition	141
8.1	Scour profile prior to a vertical weir	141
8.2	Scour at a vertical-wall abutment	144
8.3	Scour patterns at a cylindrical pier	150
8.4	Scour hole profiles at a cylindrical pier	153
8.5	Conclusions	162
9.	Conclusions	165

9.1	Incipient motion criteria	165
9.2	Adaptation of CFX-V4.3 to simulate open channel flow	166
9.3	Predictions of bead movement	167
9.4	Prediction of scour profiles	168
9.5	Limitations	169
10.	Recommendations for future research	170
	References	171
	Appendix	
A1	Subroutine USRTRN for velocity gradients between parallel plates	A1
A2	Subroutine USRTRN for the power dissipation function and the Movability Number	A5
A3	Subroutine USRBCS for the inlet velocity distribution	A7
A4	Subroutine USRWTM for the incorporating roughness effects	A12
A5	Subroutine USRBCS for setting a moving wall	A16
A6	Subroutine USRBCS for calculating k and ε distributions	A17
A7	Subroutine USRTRN for calculating the bedform critical function	A18

List of Figures

- Figure 2.1 Co-ordinate system
- Figure 2.2 Shear stress on an element
- Figure 2.3 Laminar and turbulent stress contributions with depth
- Figure 2.4 Turbulent viscosity distribution with depth
- Figure 2.5 Combined free and forced vortex (Roberson and Crowe 1997)
- Figure 2.6 Velocity distribution near a solid wall (Versteeg and Malalasekera 1995)
- Figure 2.7 Turbulent flow over smooth and rough beds (Nezu and Nakagawa 1993)
- Figure 2.8 Roughness function B_r for Nikuradse's (1933) sand roughness (Schlichting 1979)
- Figure 3.1 Settling velocity curves (Rooseboom 2001)
- Figure 3.2 Shields' (1936) diagram modified by Rouse (1939)
- Figure 3.3 Erosion and deposition criteria for uniform particles (Hjulstrom 1935)
- Figure 3.4 Velocity and shear stress distribution across an element
- Figure 3.5 Variation in applied power and power made available with depth (Rooseboom 1974)
- Figure 3.6 Critical conditions for cohesionless sediment particles (Rooseboom 1992)
- Figure 3.7 Dimensions of a small fluid element
- Figure 3.8 Stress components in the x-direction
- Figure 3.9 Components of the heat flux vector
- Figure 4.1 Scour depth at a pier as a function of time (Raudkivi 1998)
- Figure 4.2 Virtual eddy prior to a vertical weir
- Figure 4.3 Flow pattern around a submerged vertical weir (Hoffmans and Verheij 1997)
- Figure 4.4 Flow pattern at a cylindrical pier (Raudkivi 1998)
- Figure 4.5 Flow patterns around an abutment

- Figure 5.1 Degree of modelling verses computational expenditure
- Figure 6.1 Mesh refinement around a particle and in an area of interest
- Figure 6.2 Flowchart of subroutine for velocity gradients
- Figure 6.3 Velocity and velocity gradient distribution between parallel plates
- Figure 6.4 The inlet velocity distribution subroutine USRBCS
- Figure 6.5 Comparison of CFX velocity profile to experimental data
- Figure 6.6 Turbulent viscosity variation with depth
- Figure 6.7 Rigid lid model
- Figure 6.8 Moving wall model
- Figure 6.9 Moving wall surface patch routine
- Figure 6.10 Turbulent viscosity distribution with depth for moving wall and symmetry patch condition
- Figure 6.11 Revised shear stress profile – setting k and ε
- Figure 6.12 Lid-driven cavity
- Figure 6.13 Velocity vectors in a lid-driven cavity using the QUICK scheme
- Figure 6.14 The u -velocity variation with depth for a lid-driven cavity
- Figure 6.15 v -velocity distribution for various turbulent intensities
- Figure 6.16 v -velocity distribution for various dissipation length scales
- Figure 6.17 Two-dimensional flow over bedforms
- Figure 6.18 Mesh refinement around the change in geometry and section A-A
- Figure 6.19 Variation of dimensionless velocity $\frac{u}{u_*}$ with dimensionless depth $\frac{y}{D}$
- Figure 6.20 Variation in dimensionless shear stress $\frac{\overline{u'v'}}{u_*^2}$ with dimensionless depth $\frac{y}{D}$
- Figure 6.21 Subroutine for determining the topography adaptation incipient motion function
- Figure 6.22 Final methodology flowchart
- Figure 7.1 Offerman's (1998) laboratory flume
- Figure 7.2 Typical dimensions of the flow domain prior to a vertical weir
- Figure 7.3 Vector plot of velocity prior to the weir

- Figure 7.4 Unit stream power distribution prior to the weir
- Figure 7.5 Eddy prior to the suspended sill (Offerman 1998)
- Figure 7.6 Typical dimensions of the underflow structure geometry
- Figure 7.7 Flow patterns around the underflow structure
- Figure 7.8 Vector plot of velocities in the vicinity of the underflow structure
- Figure 7.9 Velocity distributions with depth
- Figure 7.10 Unit stream power distribution in the vicinity of the underflow structure
- Figure 7.11 Vortex development around a contraction (Feuilloley 1999)
- Figure 7.12 Typical dimensions of the symmetrical contraction geometry
- Figure 7.13 Velocity magnitude distribution at the channel bed and at the height of the weir for the contraction
- Figure 7.14 Flow patterns in the vicinity of the contractions
- Figure 7.15 Velocity distributions across the boundary layer eddy
- Figure 7.16 Unit stream power distribution at the channel bed and at the weir height for the contraction
- Figure 7.17 Vortex development around an expansion (Lyons 1999)
- Figure 7.18 Geometry of the expansion for the numerical model
- Figure 7.19 Velocity distribution at the sudden expansion
- Figure 7.20 The velocity vectors through the expansion
- Figure 7.21 Unit stream power distribution at the expansion and in the vicinity of the weir
- Figure 8.1 The before and after experimental sediment profiles for the vertical weir (Harnett 1998)
- Figure 8.2 Unit stream power distribution for the initial and final condition above the vertical weir
- Figure 8.3 The Movability Number for the initial and final states above the vertical weir
- Figure 8.4 Mitchell's (2000) experimental layout
- Figure 8.5 The dimensioned model for the abutment
- Figure 8.6 Velocity vectors around the abutment at mid-depth
- Figure 8.7 Total velocity across the abutment eddy at section A-A
- Figure 8.8 Velocity distribution across the slowly rotating eddy at section B-B

- Figure 8.9 Power distribution around the abutment
- Figure 8.10 Movability Number distribution around the abutment
- Figure 8.11 The pier geometry
- Figure 8.12 Velocity vectors at the pier
- Figure 8.13 Unit stream power distribution around the pier
- Figure 8.14 Movability Number distribution around the pier
- Figure 8.15 A typical self-formed channel (Babaeyan-Koopaei 1996)
- Figure 8.16 A typical channel with piers (Babaeyan-Koopaei 1996)
- Figure 8.17 The model geometry for the initial pre-scour condition
- Figure 8.18 The critical incipient motion function for each geometry nodal value
- Figure 8.19 The Movability Number distribution in the vicinity of the pier at a depth of 0.5 mm for the initial condition
- Figure 8.20 The scour potential distribution around the pier at a depth of 0.5 mm for the initial condition
- Figure 8.21 The model geometry for the final equilibrium scour condition
- Figure 8.22 Plan view of the mesh refinement in the vicinity of the pier
- Figure 8.23 Movability Number distribution for the final condition around the pier
- Figure 8.24 Distribution of the scour potential adjacent to the pier for the final equilibrium profile

List of Symbols

English alphabet:

A	cross-sectional channel area	(m ²)
A_c	Area of smallest cell	(m ²)
B	channel width	
B_r	roughness coefficient	
b	pier width	(m)
C_D	drag coefficient	
c	specific heat	(J/kg.K)
D	hydraulic mean depth	(m)
D_*	dimensionless particle size	
d	particle dimension	(m)
d_{xx}	sieve size through which xx % of particle pass by weight	(m)
e	turbulent intensity	
E_l	log-layer coefficient	
E	energy	(m)
Fr	Froude number ($= QB/Ag^{0.5}$)	
g	gravitational acceleration vector	(m/s ²)
g	gravitational acceleration scalar	(m/s ²)
h	abutment height	(m)
i	internal energy	(J)
k	turbulent kinetic energy	(m ² /s ²)
K	volume coefficient	
k_β	longitudinal bed slope factor	
k_γ	transverse bed slope factor	
k_r	bedform height	(m)
k_s	roughness height	(m)
k_s^+	dimensionless roughness	
L	turbulent dissipation length scale	(m)
ℓ	Prandtl's mixing length	(m)
n	normal vector to the channel bed	

P	pressure	(N/m ²)
$\bar{P}$	average pressure	(N/m ²)
P'	fluctuating pressure component	(N/m ²)
Pe	Peclet Number	
P_t	applied stream power	(W/m ³)
$\bar{P}_t$	average stream power	(W/m ³)
P'_t	fluctuating stream power component	(W/m ³)
P_r	particle power requirement	(W/m ³)
Q	flow rate	(m ³ /s)
Q	heat flux	(J)
q_x	x-direction heat flux component	(J)
q_y	y-direction heat flux component	(J)
q_z	z-direction heat flux component	(J)
R	eddy radius	(m)
Re	Reynold's number ($= uD/\nu$)	
Re_*	particle Reynold's number	
S_E	energy source term	
S_f	friction slope	
S_i	internal energy source term	
S_M	source term for body force effects	(kg/ms)
S_o	longitudinal bed slope	
SF	shape factor	
T	temperature	(°C/K)
t	vector $\perp$ to the velocity vector in the plane of the bed	
t	time	(s)
t_a	abutment thickness	(m)
U	velocity vector	(m/s)
$\bar{U}$	average velocity	(m/s)
U'	fluctuating velocity component	(m/s)
u	x-direction velocity component	(m/s)
$\bar{u}$	average x-direction velocity component	(m/s)
u'	fluctuating x-direction velocity component	(m/s)

u^+	dimensionless velocity	
$\tilde{u}_*$	shear/friction velocity	(m/s)
$\mathbf{u}_*$	shear/friction velocity vector	(m/s)
V	volume	(m ³)
V_{ss}	particle settling velocity	(m/s)
v	y-direction velocity component	(m/s)
$\bar{v}$	average y-direction velocity component	(m/s)
v'	fluctuating y-direction velocity component	(m/s)
w	z-direction velocity component	(m/s)
$\bar{w}$	average z-direction velocity component	(m/s)
w'	fluctuating z-direction velocity component	(m/s)
x	longitudinal flow direction	(m)
Y	potential energy per unit weight of water	(m)
y	flow depth direction	(m)
y^+	dimensionless depth	
y_0	depth at which velocity is mathematically zero	(m)
y_s	scour depth	(m)
z	transverse flow direction	(m)

Greek alphabet:

β	angle of velocity vector with longitudinal bed slope	(Degrees)
Γ	coefficient of diffusion	
γ	angle of velocity vector with transverse bed slope	(Degrees)
Δ	relative submerged density	
ε	rate of viscous dissipation	(m ² /s ³)
η	constant of proportionality	
θ	particle entrainment	
κ	von Karman constant	
λ	coefficient of bulk viscosity	(Ns/m ²)
μ	total dynamic viscosity	(Ns/m ²)
μ_t	laminar dynamic viscosity	(Ns/m ²)

μ_t	turbulent dynamic viscosity	(Ns/m ²)
ν	kinematic viscosity	(m ² /s)
ξ	linear sub-layer thickness	(m)
ρ	density of water	(kg/m ³)
ρ_s	density of sediment	(kg/m ³)
τ	shear stress	(N/m ²)
τ_b	bed shear stress	(N/m ²)
τ_{cr}	critical shear stress	(N/m ²)
Φ	dissipation function	(W/m ³)
ϕ	property per unit mass of a particle	
ϕ_s	angle of repose	(Degrees)
ψ	slope factor	
ω_b	angular velocity	(rad/s)
ω	turbulence frequency ($\omega = \varepsilon/k$)	
Ω	scour potential	

List of Tables

Table 3.1	Angle of repose for different sediment types (van Rijn 1993)
Table 3.2	Shear velocity: settling velocity ratio of various studies
Table 3.3	Summary of the Movability Numbers
Table 6.1	Default variable solvers
Table 7.1	Description of Offerman's (1998) beads
Table 7.2	Experiment and numerical model data for Offerman's (1998) experiments
Table 7.3	Maximum power value and linking power zone at the weir
Table 7.4	Description of Offerman's (1998) floatable beads
Table 7.5	Experiment and numerical model data for Offerman's (1998) experiments
Table 7.6	Maximum power values at the surface and below the structure
Table 7.7	Description of Feuilolley's (1999) beads
Table 7.8	Experiment and model data for the six contractions
Table 7.9	Maximum power at the vena contracta and over the weir
Table 7.10	Description of Lyon's (1999) beads

Chapter 1

Introduction

Scour profiles around engineering structures in open channel flow are an essential parameter in foundation design. Excessive scour around structures such as bridge piers can progressively undermine the foundations resulting in possible structural failure and collapse. Engineering structures such as weirs, sills, piers and abutments obstruct the flow in open channel flow, resulting in steep velocity gradients, a higher degree of turbulence and hence a stronger erosive capacity in the water. Ultimately they cause scour of the channel bed. Consequently it is necessary to predict scour hole profiles and maximum scour depths to prevent foundation failure. This thesis investigates a computational fluid dynamics (CFD) model based on the unit stream power approach for the prediction of scour profiles around hydraulic structures.

Current scour prediction methods typically include empirically based equations relating to the geometry of the structure. Alternatively, small-scale hydraulic models are used. Incipient motion criteria for the initiation of sediment movement have largely been based on a critical velocity approach (Fortier and Scobey 1926; Hjulstrom 1935) or alternatively a critical bed shear stress (Shields 1936). Bagnold (1960) first introduced the concept of stream power as an alternative approach to predicting sediment transport. Yang (1972) defined the unit stream power as the product of the velocity and the slope i.e. the release of potential energy. Ackers and White (1973) defined the dimensionless sediment transport parameter based on the effective stream power for the bedload component and the total stream power for the suspended load. This unit stream power was interpreted as the product of the bed shear stress and the velocity i.e. the energy expended in overcoming the resistive force. Rooseboom (1974) defined the unit stream power distribution with particular emphasis given to the region adjacent to the channel bed, where the applied stream power is at a maximum. Yang and Song (1979) showed that the unit stream power at a point in the flow is identical to the dissipation function for three-dimensional incompressible flow due to gravitational acceleration in terms of the local shear stresses, velocity gradients and turbulent viscosity. This forms the fundamental applied stream power equation throughout this study. The applied stream power can

be compared to a particle's independent power requirement (Rooseboom 1974) to determine whether particle movement will occur.

This applied stream power can alternatively be expressed in terms of the ratio of the shear velocity at the channel bed to the settling velocity of a particle (Rooseboom 1974; Armitage 2001) defined as the Movability Number (Liu 1957). This is a common criterion for incipient motion (Chanson 1999) and provides a useful comparison for the first approach. Furthermore, these criteria can be adapted to incorporate the angle of repose of the sediment for areas where steep topography predominates. A factor is introduced for the Movability Number that accounts for the angle of repose as well as the longitudinal and transverse directions of the flow to the horizontal plane (Armitage 2001).

The difficulty in implementing Yang and Song's (1979) equation is in accurately determining the localised velocity gradients, shear stresses and turbulent viscosity. This study introduces a CFD model for the solution of these flow variables. A commercially available CFD software, CFX-V4.3 (1999), which is based on the finite volume method was utilised. This version of CFX did not incorporate an algorithm for modelling free surfaces and had to be adapted to simulate open channel flow. A large component of this thesis involves this adaptation combined with the development of additional programs that were written to solve for variables such as the stream power. CFX-V4.4 (2001) was released in the concluding stages of this research period and was implemented for the final, most complex application.

The stream power profiles generated by the numerical model were tested against experimental data of bead movement and observed scour profiles around hydraulic structures.

Offerman (1998), Feuilloley (1999) and Lyons (1999) researched the movement of beads around engineering structures in open channel flow, using small-scale physical models. The unit stream power theory was implemented to predict the movement of beads over a vertical weir (Offerman 1998), through a sudden contraction (Feuilloley 1999) and through a sudden expansion (Lyons 1999). The movement of 'floatables', i.e. particles less dense than water, around an inverted weir or 'underflow structure'

was also investigated (Offerman 1998). The initial part of this study involves generating unit stream power distributions with the CFD software and comparing these with the bead movement that was observed in the laboratory. The prediction of the bead paths forms the basis upon which it was possible to predict the movement of more refined particles, i.e. sand, around hydraulic structures.

The movement of sand particles is more complex and requires consideration of boundary layer development, hydraulically rough or hydraulically smooth conditions, armouring effects and the precise determination of the settling velocities. The numerical model was tested against the specific case of local scour around hydraulic structures such as weirs, abutments and piers. The CFD software was used to generate power distributions that predict the scour profiles adjacent to these engineering structures. These distributions were then compared to observed scour profiles from existing laboratory data (Babaeyan-Koopaei 1996; Harnett 1998; Mitchell 2000; Midgley 2000).

Distributions of Movability Numbers were also generated as an alternative scour prediction method and compared to the power profiles. These distributions were further adapted for steep topography in the region immediately adjacent to the abutments and piers.

The thesis is structured as follows:

Chapter 2 provides a description of the properties of water and open channel flow with particular reference to boundary layer theory and the impact of roughness. It is necessary in the adaptation of CFX-V4.3 (1999) for open channel flow to describe the characteristics of shear stress, turbulent viscosity, velocity distribution as well as concepts such as vortex development. Vortex street development typically occurs at the separation point of a hydraulic structure e.g. an abutment or pier. Eddies and virtual eddies are also located in the vicinity of the structure e.g. the slowly rotating eddy downstream of an abutment.

Chapter 3 reviews the previous methods of predicting sediment movement. The basic sediment characteristics such as particle size, shape, density, angle of repose and the

settling velocity are introduced, all of which are essential to determining the particle power requirement. The unit stream power approach as developed initially by Bagnold (1960) is introduced, followed by the two-dimensional unit stream power distribution (Rooseboom 1974) and finally a three-dimensional approach (Yang and Song 1979). The relationship between the applied unit stream power and the Movability Number (Liu 1957) is outlined and this relationship is adapted to include a factor to account for the local topography (Armitage 2001).

Chapter 4 introduces typical flow patterns around engineering structures such as weirs, abutments and piers that result in the respective scour profiles. Particular emphasis is placed on the vortex development, as the steep velocity gradient across these vortices contributes to the overall power input. Previous numerical modelling approaches to scour prediction are briefly reviewed.

Chapter 5 provides an introduction to the finite volume method and the resulting choices that are encountered in its application. Concepts such as differencing schemes, solvers and turbulence models are discussed with reference to the available options in CFX-V4.3 (1999).

Chapter 6 describes the adaptation of CFX to simulate open channel flow characteristics, including the additional subroutines that were incorporated. A detailed description of the options selected from those available (outlined in Chapter 5) is provided. Finally a detailed methodology is presented by means of a flow chart for the implementation of each application of the adapted numerical model.

Chapter 7 reviews the results for the prediction of bead movement around vertical weirs, suspended sills, contractions and expansions. Power distributions and flow fields are generated and compared to observed behaviour in the laboratory.

Chapter 8 presents the predictions of the scour patterns around a vertical weir, a vertical-wall abutment and a cylindrical pier. These are based on the applied stream power distributions and the Movability Number. The scour hole profile adjacent to a cylindrical pier centrally located in a self-formed channel is predicted. This

prediction is based on the Movability Number that has been adapted to account for local topography in the vicinity of the structure.

Chapter 9 provides a discussion on the results of the predicted bead movement and sediment scour around the various hydraulic structures. The difficulties encountered with CFX-V4.3 (1999) and CFX-V4.4 (2000) are briefly discussed.

Chapter 10 includes recommendations for future research.

University of Cape Town

Chapter 2

Open Channel Flow Characteristics

Simulation of an alluvial channel by means of a numerical model necessitates a comprehensive understanding of the fluid and sediment properties. Water is the fluid that forms the basis for this study. This chapter describes the fundamental fluid properties of water, the open channel flow characteristics and the measurements that are typically used to portray the approximate channel dimensions, all of which are referred to throughout this document.

2.1 Co-ordinate System

The co-ordinate system is defined below (fig 2.1). The flow occurs in the x -direction with the channel width and depth in the z - and y -direction respectively. The velocity vector U and its components u , v and w are as shown in the diagram.

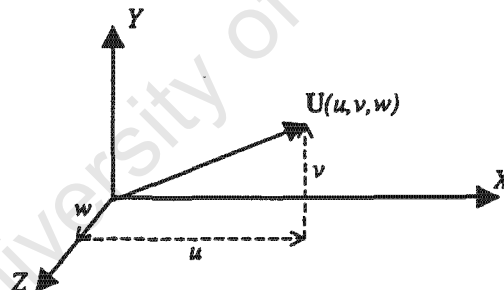


Figure 2.1 : Co-ordinate system

2.2 Density, Shear Stress and Viscosity

Density ρ varies with temperature and pressure, however for this study it is assumed to be 1000 kg/m^3 (Chadwick and Morfett 1993; Raudkivi 1998).

The shear stress therefore increases linearly from zero at the surface to a maximum value of $\bar{\rho}gS_oD$ at the bed. The velocity distribution can only be given a mathematical function if the relationship between shear stress τ and the velocity gradient (du/dy) is known. The laminar dynamic viscosity μ_l (Ns/m²) is the constant of proportionality relating the shear stress and the velocity gradient, du/dy (s⁻¹), in a Newtonian fluid (Schligting 1979; Chadwick and Morfett 1993; Roberson and Crowe 1997) such that,

$$\tau = \mu_l \frac{du}{dy}$$

2.3

The laminar dynamic viscosity is dependent on the temperature T and may be approximated by (Raudkivi 1998),

$$\mu_l = \frac{1.79 \cdot 10^{-3}}{1 + 33.68 \cdot 10^{-3}T + 2.21 \cdot 10^{-4}T^2}$$

2.4

It is approximately 1.0×10^{-3} Ns/m² at 20 °C. The kinematic viscosity ν (m²/s) is the ratio between the dynamic viscosity and the fluid density,

$$\nu = \frac{\mu}{\rho}$$

2.5

2.3 Reynolds Stress Model

Turbulent flow occurs at high Reynolds Numbers ($Re > 2000$) when the flow incorporates an eddying or mixing action and the inertia forces dominate the viscous forces. The fluid particles are therefore continuously interchanging and it follows that the fluid properties may be exchanged. The Reynolds' stress model incorporates the fluctuation of the velocity components and is given as (Chadwick and Morfett 1993),

$$\tau = \rho \overline{v'u'}$$

2.6

which is termed a Reynolds' Stress. For three-dimensional flow, the symmetrical Reynolds' stress tensor is given by (Schlichting 1979; Versteeg and Malalasekera 1995)

$$\tau_{ij} = \begin{pmatrix} \overline{\rho u'^2} & \overline{\rho u'v'} & \overline{\rho u'w'} \\ \overline{\rho u'v'} & \overline{\rho v'^2} & \overline{\rho v'w'} \\ \overline{\rho u'w'} & \overline{\rho v'w'} & \overline{\rho w'^2} \end{pmatrix}$$

2.7

where the suffix j is the direction of the stress component and the suffix i is the normal to the surface that the stress acts on and $\overline{\rho u'^2}$, $\overline{\rho v'^2}$ and $\overline{\rho w'^2}$ are the normal stresses.

Boussinesq (1877) proposed that with the onset of turbulence, the Reynolds stresses could be related to the mean rate of deformation by a turbulent or eddy viscosity μ_t . The total shear stress would hence be given by the laminar and turbulent components (fig 2.3),

$$\tau = \mu \frac{du}{dy} = (\mu_l + \mu_t) \frac{du}{dy}$$

2.8

μ = total dynamic viscosity (Ns/m²)

or in suffix notation (Versteeg and Malalasekera 1995),

$$\tau_{ij} = -\rho \overline{U_i' U_j'} = (\mu_l + \mu_t) \left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]$$

2.9

where $\underline{U}' = (u', v', w')$. Figure 2.4 shows the turbulent viscosity distribution with depth (Vanoni 1946; Schlichting 1979; Nezu and Nakagawa 1993).

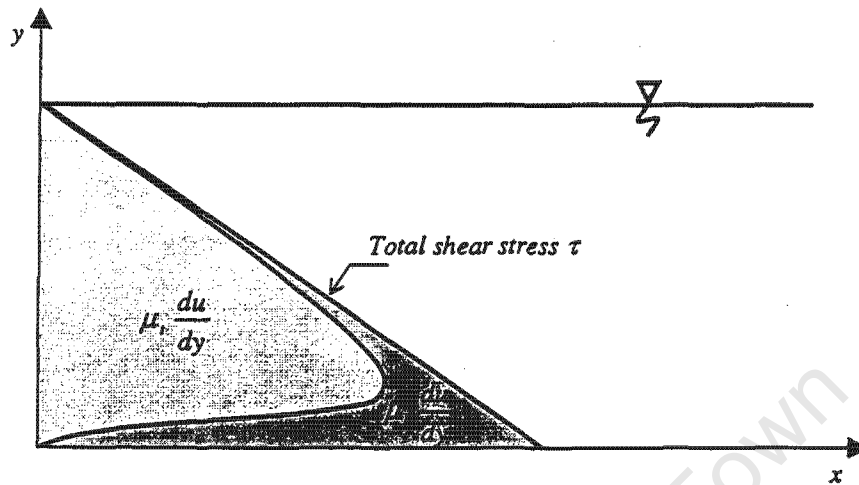


Figure 2.3 : Laminar and turbulent stress contributions with depth

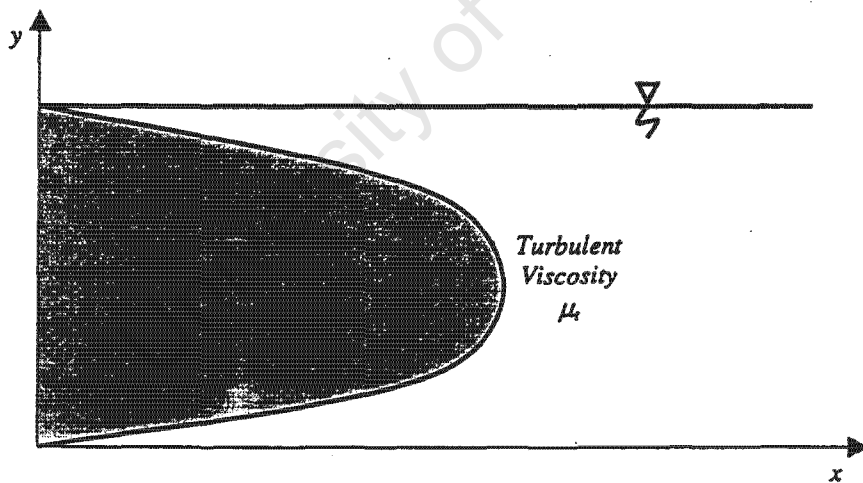


Figure 2.4 : Turbulent viscosity distribution with depth

The laminar viscosity varies with temperature and is approximately equal to 0.001 Ns/m^2 at 20°C (Section 2.2).

2.4 The Prandtl Eddy Model

Prandtl (1925) defined the shear stress for turbulent flow as a function of the velocity gradient across an eddy and the eddy viscosity μ_t , such that

$$\tau = \mu \frac{du}{dy} = (\mu_l + \mu_t) \frac{du}{dy} \quad 2.8$$

where the total dynamic viscosity could be given by

$$\mu_t = \rho \ell^2 \frac{du}{dy} \quad 2.10$$

and ℓ is a mixing length. For uniform flow, he found that the mixing length ℓ is proportional to the radius of the eddy or the distance a fluid particle may randomly travel in a turbulent flow. The mixing length is related to the distance from the boundary by $\ell = \kappa y$ (Schlichting 1979; White 1991) where y is the depth and κ is the von Karman coefficient. Hence equation 2.8 can be written as,

$$\tau = \mu_l \frac{du}{dy} + \rho \kappa^2 y^2 \left(\frac{du}{dy} \right)^2 \quad 2.11$$

κ is approximately $(1/2\pi)^{0.5}$ (Rooseboom 1992) or 0.4 (Millikan 1939; Versteeg and Malalasekera 1995) for clear sediment-free fluids. Hinze (1962) found that the von Karman constant in clear water varies from 0.36 to 0.43 over a range of Reynolds Numbers. The uncertainty in κ of $\pm 10\%$ is due to an uncertainty of $\pm 20\%$ of the value of the boundary stress (Long et al. 1993). Long et al. (1993) evaluated κ from integral flow parameters and the analysis yielded $\kappa = 0.408 \pm 0.004$. κ decreases in sediment laden fluids. Refer to Rodi and Celik (1984) for additional reading on the definition of κ based on the velocity profiles.

Prandtl's (1925) formula is limited for use with alluvial channel flow, as it assumes the sediment concentration is dilute and it neglects the effect of density stratification. Furthermore, it ignores the fact that there are two distinct phases in sediment-laden flow. Many researchers have addressed these drawbacks of Prandtl's (1925) formulation. This formula has hence been modified to account for equilibrium sediment laden flows. See further details in Kovacs (1998).

2.5 Free, Forced and Rankine Vortices

Turbulent flow is characterised by an eddying or mixing action, which results in a transfer of momentum between the fluid particles. This eddying phenomenon has been modelled (Section 2.3 and 2.4) to determine the shear stresses. This section introduces the nature of the fluid motion within the eddies.

When flow patterns follow a circular path with streamlines as concentric circles, the flow is termed a "vortex". Vortices can be forced, free or a combination of these which is termed a Rankine vortex.

A forced vortex occurs when the flow increases linearly from the centre of rotation. The fluid motion approximates that which is generated when a mechanical rotor acts on a fluid. A free or potential vortex is a naturally occurring circular flow such as water down a plug hole or around a river bend. The product of the velocity at a point and the radial distance from the vortex centre to that point is a constant, so the velocity at the centre theoretically tends to infinity. Certain naturally occurring vortices such as tornadoes or whirlpools in rivers are a combination of the free and forced vortex (fig 2.5).

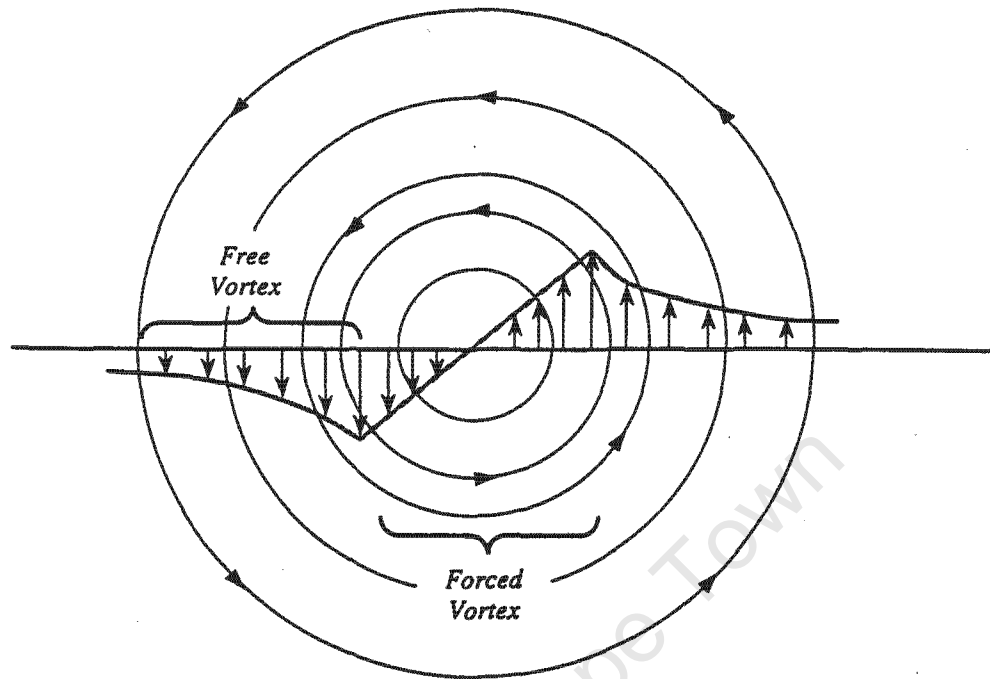


Figure 2.5 : Combined free and forced vortex (Roberson and Crowe 1997)

2.6 The Boundary Layer

In uniform open channel flow, the boundary layer is fully developed and extends from the channel bed throughout the flow depth. Adjacent to the bed is the laminar sub-layer or viscous layer. As the distance from the bed increases, the viscous effects are reduced and the inertia forces increase. This relatively small region is termed the transitional zone and at a greater distance from the bed is the turbulent zone where the inertia forces are dominant

In the laminar sub-layer it can be shown that,

$$\tau = \mu_l \frac{du}{dy}$$

2.3

here τ is virtually constant and equal to the shear stress τ_o at the wall. Since du/dy is also nearly constant adjacent to the channel bed,

$$\frac{du}{dy} = \frac{\tau_o}{\mu_l} \quad 2.12$$

Integrating with respect to depth,

$$u = \frac{\tau_o y}{\mu_l} \quad 2.13$$

which gives a linear velocity distribution in the viscous layer. The shear velocity u_* (also known as the friction velocity) is defined as,

$$u_* = \sqrt{\frac{\tau_o}{\rho}} \quad 2.14$$

Dividing 2.13 by the density and substituting equation 2.14 yields,

$$\frac{u}{u_*} = \frac{y u_*}{\nu} \quad (\text{laminar}) \quad 2.15$$

which describes the velocity distribution in the laminar region. In the turbulent region the stress is given by,

$$\tau = \mu_l \frac{du}{dy} + \rho \kappa^2 y^2 \left(\frac{du}{dy} \right)^2 \quad 2.11$$

Dividing through by the density, substituting equation 2.14 (for the shear velocity) and integrating yields,

$$\frac{u}{u_*} = \frac{1}{\kappa} \ln y + B_r \quad (\text{turbulent})$$

2.16

which is only applicable for unidirectional flow. This equation describes the velocity distribution in the transitional and turbulent layers. The constant B_r is dependent on the surface roughness k_s which can be determined for different materials, and the bedform height k_r . $B_r = 5.5$ (Schlichting 1979; Versteeg and Malalasekera 1995) for hydraulically smooth conditions.

A dimensionless velocity u^+ and depth y^+ can be defined as (Versteeg and Malalasekera 1995),

$$u^+ = \frac{u}{u_*}$$

2.17

$$y^+ = \frac{u_* y}{\nu}$$

2.18

Substituting these into equations 2.15 and 2.16 which describe the velocity distributions for the laminar and turbulent layers respectively, yields (Nezu and Nakagawa 1993; Versteeg and Malalasekera 1995)

$$u^+ = y^+ \quad y^+ < 5$$

2.19

$$u^+ = \frac{1}{\kappa} \ln y^+ + B_r = \frac{1}{\kappa} \ln E_l y^+ \quad 70 < y^+ < 500$$

2.20

E_l is the log layer coefficient and is a universal constant for all turbulent flows past smooth walls at high Reynolds Numbers. $E_l = 9.8$ (Versteeg and Malalasekera 1995)

for hydraulically smooth conditions. The linear sub-layer occurs where $y^+ < 5$ and the linear sub-layer thickness ξ is given by,

$$\xi = \frac{5\nu}{u_*}$$

2.21

The area for $5 < y^+ < 70$ is known as a 'buffer layer' and no simple velocity formula is available for this region. Spalding (1961) has developed a single composite formula that combines the linear and logarithmic distributions for this region.

The onset of turbulence occurs for y^+ values of ± 5 . At $70 < y^+ < 500$ the flow is considered fully turbulent and the velocity distribution is governed by equation 2.20, commonly termed 'the law of the wall'. Figure 2.6 shows the variation of u^+ with $\log y^+$. The grey band indicates the range of experimental values (Schlichting 1979). The two functions intersect at some point in the transitional region. The nature of this transitional curve is undefined. The intercept is usually determined by solving equations for the smooth and rough boundaries simultaneously. The values of this intercept range between $y^+ = 11$ and $y^+ = 13$ (Yalin 1972; Rooseboom 1992). The most widely used value is $y^+ \approx 11.8$ (CFD codes use 11.84 – CFX Manual, 1999). For rough surfaces $y^+ = 11.6$ (Einstein 1950; Raudkivi 1998).

At great depths, i.e. $y^+ > 500$, the flow is less influenced by the boundary. The flow in this region is free of shear and therefore not subjected to viscosity related forces. Cole's (1956) law of the wake can be used to describe the velocity distribution in this outer layer.

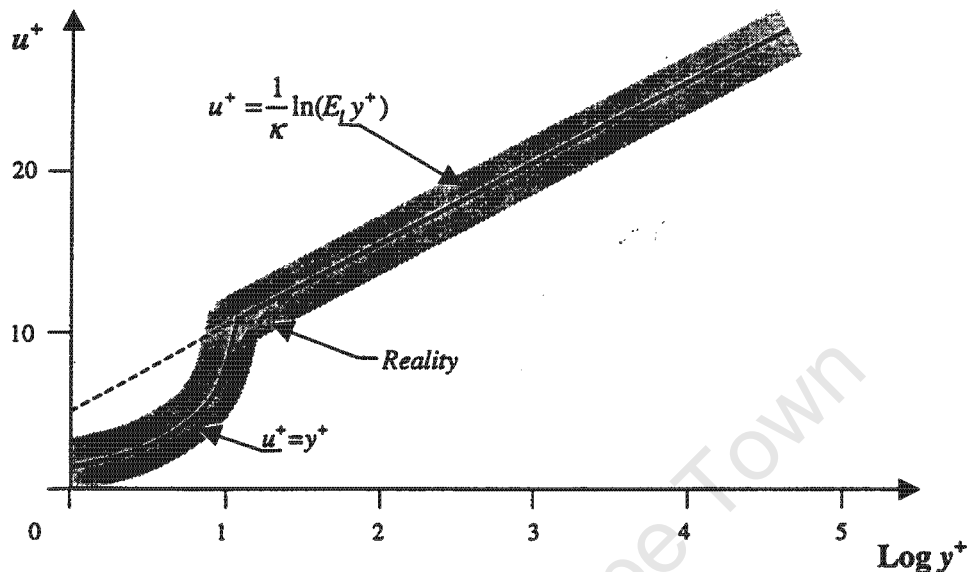


Figure 2.6 : Velocity distribution near a solid wall (Versteeg and Malalasekera 1995)

2.7 Effect of Roughness on the Boundary Layer

The effect of roughness on the velocity distribution is paramount to the study of local scour at the channel bed. These channel beds are composed of sand grains that can have complex bed configurations such as ripples and dunes. The size of the roughness elements and the height at which the theoretical wall level should be located i.e. where $y = 0$ are the two principal parameters in determining the roughness coefficient for the velocity distribution.

Nikuradse (1933) used equivalent sand roughness k_s for his systematic experiments in pipe flows. For a rough bed, composed of uniform sand grains, he defined the sand diameter as k_s . Since both the shear velocity and the roughness effect the flow in the boundary layer, a dimensionless roughness k_s^+ can be defined where,

$$k_s^+ = \frac{u_* k_s}{\nu}$$

2.22

such that the effect of turbulence is defined by both the shear velocity and the roughness height. The roughness can be classified into three categories (Schlichting 1979; Nezu and Nakagawa 1993)

- (i) hydraulically smooth bed $k_s^+ < 5$
- (ii) transitional bed $5 \leq k_s^+ \leq 70$
- (iii) completely rough bed $k_s^+ > 70$

Figure 2.7 shows turbulent flow over a hydraulically smooth and hydraulically rough bed respectively.

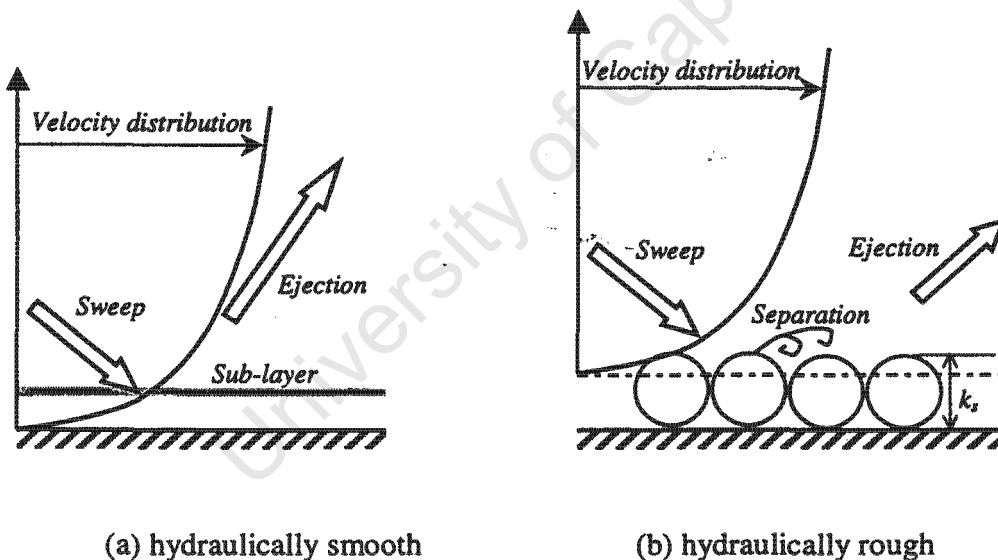


Figure 2.7 : Turbulent flow over smooth and rough beds (Nezu and Nakagawa 1993)

The roughness effects are suppressed if the bed is hydraulically smooth, as the roughness elements are buried in the viscous sub-layer. In the case of completely rough beds, the viscous effects are negligible because the roughness elements penetrate deep into the fully turbulent logarithmic layer. The transitional region

occurs where the effects of both roughness and viscosity occur. The 'law of the wall' (equation 2.20) can be rewritten to incorporate roughness

$$u^+ = \frac{1}{\kappa} \ln \left(\frac{y^+}{k_s^+} \right) + B_r = \frac{1}{\kappa} \ln \left(\frac{E_t y^+}{k_s^+} \right)$$

2.23

For the hydraulically smooth case B_r is given by,

$$B_r = 5.5 + \frac{1}{\kappa} \ln k_s^+$$

2.24

$B_r = 8.5$ (Schlichting 1979; Nezu and Nakagawa 1993) for the completely rough case. The value of B_r for the transitional case can be obtained from figure 2.8 using the data from Nikuradse's (1933) sand roughness experiments. The grey area indicates the typical experimental data scatter.

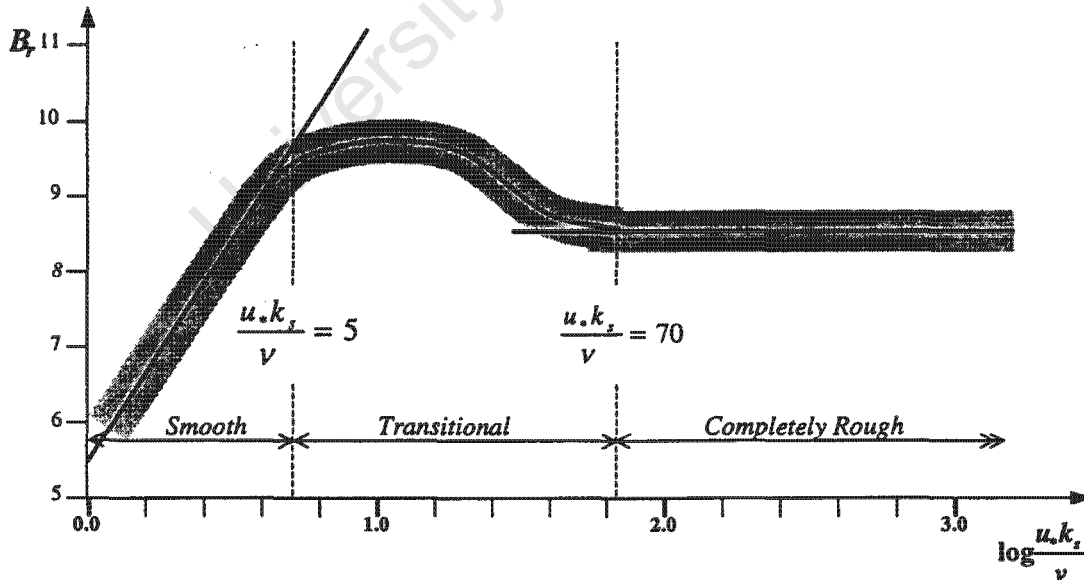


Figure 2.8 : Roughness function B_r for Nikuradse's (1933) sand roughness (Schlichting 1979)

Typical k_s values for surface roughness are available in most texts (e.g. Zegzhda 1938; Ackers 1958; Chadwick and Morfett 1993). For sand bed channels, some of the suggested values are:

$$k_s = d_{65} \quad (\text{Einstein 1950})$$

$$k_s = d_{90} \quad (\text{Meyer-Peter and Muller 1948})$$

$$k_s = d_{85} \quad (\text{Simons and Richardson 1966})$$

Where d_{xx} is the sieve size through which xx % of the particles pass by weight.

2.8 Alternative Velocity Distribution for Turbulent Flow

Prandtl's (1925) approach for describing the velocity distribution can be extended to develop an equation for the average channel velocity for the fully developed turbulent boundary layer. These equations are utilised for this study in the adaptation of the CFD model for determining the velocity at a particular depth as a function of the average velocity. The eddying motion of the fluid as described by Prandtl (1925) can be expressed as (Rooseboom 1992)

$$\frac{u}{u_*} = \frac{1}{\kappa} \ln \left(\frac{y}{y_o} \right)$$

2.25

where y_o is the channel depth at which the velocity is mathematically equal to zero. This equation is similar to 2.16 although it eliminates the constant B_r . y_o can be determined from the roughness factor k_s (Simons and Senturk 1977),

$$y_o = \frac{k_s}{29.6}$$

2.26

Integrating equation 2.25 over the depth gives the average velocity $\bar{u}$ (Rooseboom 1992),

$$\frac{\bar{u}}{u_*} = \frac{1}{\kappa} \ln \left(\frac{D}{ey_o} \right)$$

2.27

where D is the flow depth.

2.9 Conclusions

This chapter has provided the fundamental fluid properties that are referred to throughout this document. The channel flow characteristics such as:

- (i) shear stress and turbulent viscosity distribution with depth
- (ii) vortex properties
- (iii) boundary layer phenomena and the roughness effect
- (iv) the velocity distribution (equations 2.25 & 2.27) that is later used for the numerical model (Chapter 6)

are discussed in detail as they are fundamental in the understanding of the later chapters.

Chapter 3

Incipient Motion Criteria

This chapter gives a brief review of the various incipient motion criteria focusing on the unit stream power approach, which was implemented throughout this study.

3.1 The Sediment Particles

The sediment particles in this study are beads and sand. These can be defined according to the properties of a sample of representative particles in terms of their density, size and settling velocity

3.1.1 Cohesiveness

A dominant factor effecting sediment entrainment is the degree to which the particle is cohesive e.g. cohesive sands have a high clay content and electro-magnetic forces occur in the clay-water medium. The clay minerals affect the soil properties such that the particles tend to behave as one large coherent mass. It has been shown that once suspended, the cohesive mass breaks up and the particles behave as non-cohesive materials (Vanoni 1975).

The remainder of this study will only be covering non-cohesive sediment types. Non-cohesive soils form a granular structure rather than a coherent mass. These particles are treated independently with a particle size d and density ρ_s .

3.1.2 Particle Size

A number of methods are used to determine a representative particle size (details in Van Rijn 1993; Yang 1996; Raudkivi 1998). The d_{50} size is commonly used (Yang 1996; Raudkivi 1998). It is the sieve size through which 50% of the particles pass by weight.

3.1.3 Particle Density

The mineral composition and density of a particle is related to the type of rock it is derived from. In open channel flow, alluvial deposits of quartz particles, with density $\rho_s = 2650 \text{ kg/m}^3$, are dominant. The densities of the bead was dependent on the substance from which it was derived e.g. cork or polystyrene.

3.1.4 Particle Settling Velocity

The particle settling velocity is a function of the size, shape and density of the particle and the viscosity of the fluid. The settling velocity differs in moving and still water, in laminar or turbulent flow (fig 3.1) and with the number of particles falling as there is interference between particles. In turbulent flow the random up-draughts of water can further reduce the absolute settling velocity of a particle. Bagnold (1966) postulated that no particles remain in suspension unless the upward velocity of the turbulent eddies exceeds the particle settling velocity.

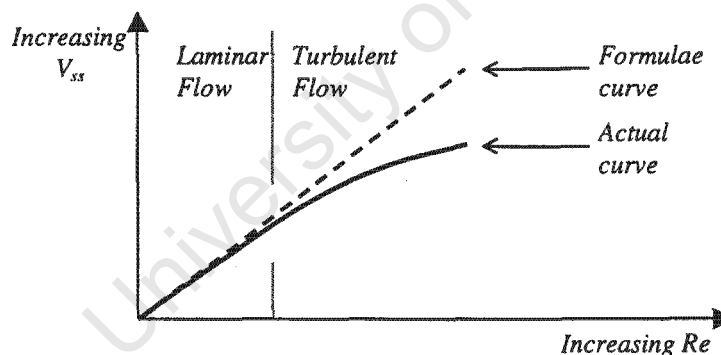


Figure 3.1 : Settling velocity curves (Rooseboom 2001)

Incipient motion of sediment particles commences at the channel bed. These particles are situated in the laminar sub-layer for the hydraulically smooth case, while extending through to the turbulent layer for the hydraulically rough case (Section 2.7).

There have been many studies into determining the fall velocity, both empirical and analytical (e.g. Zanke 1982; Cheng 1997). Two of the more widely used formulae that were implemented in this study are discussed here:

- (i) Falling under the influence of gravity the particle will reach a constant velocity, the terminal velocity, when the drag force equals the submerged weight of the particle. For a spherical particle in stationary fluid, the fall velocity V_{ss} is,

$$V_{ss} = \left[\frac{4}{3} \frac{gd}{C_D} \left(\frac{\rho_s}{\rho} - 1 \right) \right]^{0.5} \quad 3.1$$

Where C_D is the drag coefficient. $C_D \approx 1.0$ for natural coarse sand and gravel (Raudkivi 1998). Further details on determining C_D are given in Stokes (1851), Schiller and Naumann (1933), Rubey (1933), Kazanskij (1981) and Chien and Wan (1998).

- (ii) Cheng (1997) fitted a curve to published data on fall velocities.

$$\frac{V_{ss}d}{\nu} = \left[(25 + 1.2D_*^2)^{0.5} - 5 \right]^{1.5} \quad 3.2$$

V_{ss} = settling velocity (cm/s)

d = particle diameter (cm)

where

$$D_* = \left[\frac{\Delta g d^3}{\nu^2} \right]^{1/3} \quad 3.3$$

D_* = dimensionless particle size

Δ = relative submerged density given by,

$$\Delta = \frac{\rho_s}{\rho} - 1$$

3.4

3.1.5 Angle of Repose

A sediment particle is considered stable in a horizontal plane or a slope when the centre of gravity of the particle is vertically above the point of contact. The critical angle of the plane to the horizontal at which incipient motion occurs is termed the angle of repose ϕ_s . Thus a particle situated on a positive longitudinal slope may reach incipient motion conditions more quickly than a particle on a horizontal bed or adverse longitudinal slope, and thus become unstable.

d_{50} (mm)	ϕ_s rounded particles (degrees)	ϕ_s angular particles (degrees)	Description
<1	30	35	
5	32	37	gravel
10	35	40	gravel
50	37	42	gravel
>100	40	45	cobble/boulders

Table 3.1 : Angle of repose for different sediment types (van Rijn 1993)

The angle of repose is dependent on the nature of the sediment such as the particle shape, and it increases with the angularity of the sediment. Table 3.1 lists the angle of repose for various particle dimensions (van Rijn 1993), which range from 30° to 45°.

3.1.6 Application of the Sediment Properties

The properties of sediment such as the density and the settling velocity are fundamental in determining the power requirement (Section 3.3.2) for a sediment

particle. This particle power requirement is essential to the unit stream power approach to incipient motion conditions (Section 3.3). The power requirement changes for a particle situated on a slope e.g. it decreases for a particle situated on a positive longitudinal slope.

3.2 Sediment Motion

In an alluvial channel, sediment moves in three different modes; rolling/sliding, saltation and suspension (Chadwick and Morfett 1993). Rolling and sliding occur when the sediment is forced to move along the riverbed, maintaining contact with the bed. Saltation is when the particles hop along following parabolic-type trajectories. Suspension occurs when the particle is no longer in contact with bed. Generally the bed-load transport rate of a typical river is about 5-25% of that in suspension (Yang 1996).

Many theoretical and empirical studies have been undertaken to describe incipient motion conditions in uniform open-channel flow. Due to the stochastic nature of the sediment in alluvial channels, it is difficult to define precisely under what flow conditions a sediment particle will begin to move. The study of incipient motion is therefore the interpretation of these conditions.

There are three main approaches:

- (i) The critical velocity criteria. (Fortier and Scobey 1926, Hjulstrom 1935, Neill 1967, Vanoni 1975, Yang 1976 and Zanke 1977)
- (ii) The bed shear stress criteria. (Shields 1936, White 1940, Brooks 1955, Liu 1958 and Yang 1972)
- (iii) The unit stream power approach. (Bagnold 1960, Yang 1972, Ackers and White 1973, Rooseboom 1974, Annandale 1986)

Other methods include those by Meyer-Peter and Muller (1948) and Mavis and Laushey (1948) that take factors such as armouring into account.

3.2.1 The Critical Velocity Approach

The critical mean velocity is used to determine whether a sediment particle on the bed of an open channel, is ready to move i.e. when incipient motion is imminent. An example of a typical critical velocity approach is Hjulstrom's (1935) curve for the prediction of sediment movement based on the grain size and the velocity (fig 3.3). The critical mean velocity can also be calculated for given conditions from the theoretical shear stress or the shear velocity. This is then compared to the actual velocity to determine whether incipient motion conditions are imminent. Further details in Raudkivi (1998).

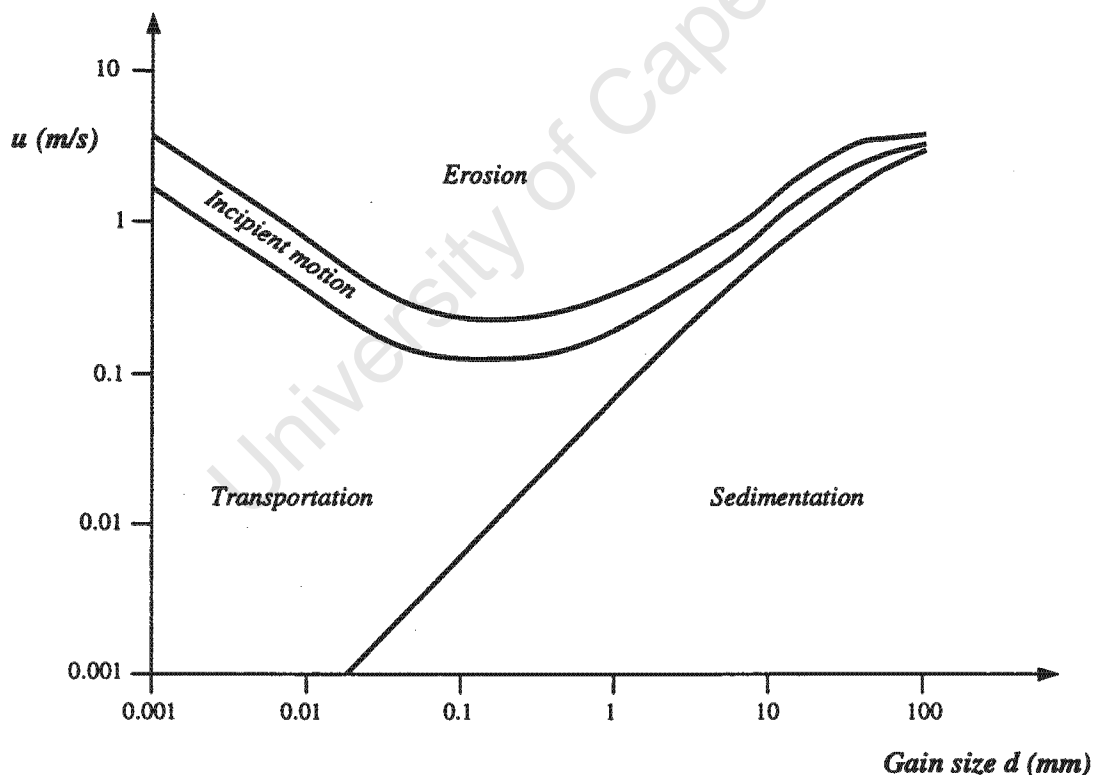


**Figure 3.3 : Erosion and deposition criteria for uniform particles
(Hjulstrom 1935)**

Determining the incipient motion criteria using the critical velocity is limited by the difficulty of measuring localised velocities in the boundary region in order to determine the drag and lift forces. The velocity gradient is greatest near the boundary and it varies with water depth and sediment roughness.

3.2.2 The Shear Stress Approach

The shear stress approach is based on an individual particle on the channel bed and the forces acting on it. A particle situated on the river floor will move forward and upward when the forces exerted on the body are not in equilibrium, causing a resultant acceleration. At the threshold of movement, the bed shear stress τ_o will equal a critical shear stress τ_{cr} . Shields (1936) investigated this and found that the particle entrainment θ , must be some function of the particle Reynolds Number Re_* , which was a dimensionless function of the density ρ , viscosity μ , the particle diameter d and the friction velocity u_* , such that

$$Re_* = \frac{\rho u_* d}{\mu}$$

3.5

The particle entrainment θ is given by

$$\theta = \frac{\tau_{cr}}{(\rho_s - \rho)gd}$$

3.6

Shields (1936) plotted θ against Re_* . Rouse (1939) later added a line to Shields' (1936) diagram that indicated the threshold of movement for a particle (fig 3.2).

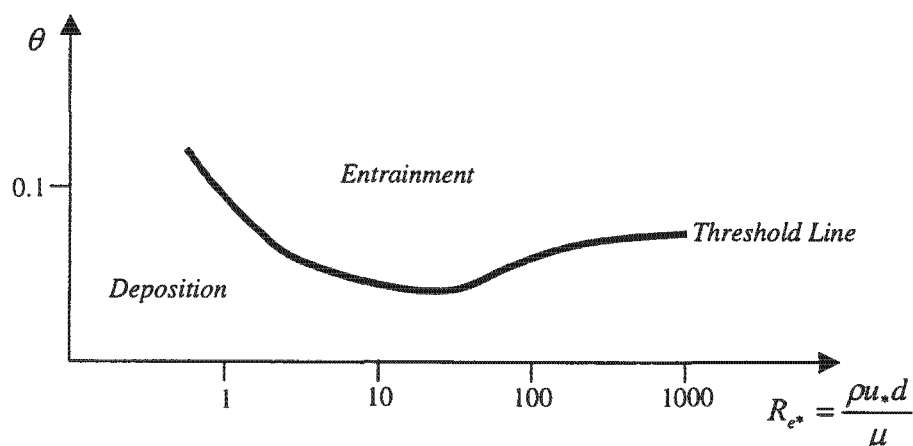


Figure 3.2 : Shields' (1936) diagram modified by Rouse (1939)

Yang (1973) pointed out seven factors that suggested Shield's diagram is not the most desirable criterion for incipient motion. The primary drawback of this method is that the particle grain size is not a truly representative measure of transportability. However, despite criticism, it is still the most widely used criterion of incipient motion.

3.3 One Dimensional Unit Stream Power Approach

Hydraulic principles and theories on the understanding of fluid mechanics, are based on the laws of mass, energy and momentum. These equations have been adequate in examining fluid properties and flow, however, in the case of sediment transport, the conservation of power has been recognised as a potentially more accurate measure of sediment movement (Yang 1972; Rooseboom 1974 & 1992). Basic principles of non-equilibrium thermodynamics may be used to show that the applied stream power approaches a constant minimum value throughout a non-equilibrium system when stable conditions are approached. Applying this concept to open channel flow, implies that an alluvial channel will adjust its velocity, slope, roughness and geometry in such a way that a minimum amount of unit stream power is used to transport a given sediment and water discharge. This hypothesis can also be used to derive a criterion where these equilibrium/incipient motion conditions are present.

3.3.1 Minimum Rate of Energy Dissipation

Bagnold (1960) first introduced the concept of stream power as an alternative approach to predicting sediment transport. Yang (1972) defined the unit stream power as the product of the velocity and the slope i.e. the release of potential energy. The theory of minimum rate of energy dissipation states that when a dynamic system reaches its equilibrium condition, its rate of energy dissipation is at a minimum. For a uniform flow of given channel width, neglecting any energy dissipation due to sediment transport, the rate of energy dissipation per unit weight of water is,

$$\frac{dY}{dt} = \frac{dx}{dt} \frac{dY}{dx} = VS$$

3.7

Y = potential energy per unit weight of water

V = average velocity

S = average slope

VS = unit stream power

3.3.2 Stream Power Distribution in an Infinitely Wide Channel

Rooseboom (1974) suggested the unit stream power approach to predict individual particle movement. The principle of this approach is that it is possible to determine the power dissipation intensity at a point in a water column and hence compare it to the amount of power necessitated to cause particle suspension. These powers may be related by a constant of proportionality, η (Armitage 2001). Should the stream power capacity exceed the power required for entrainment multiplied by η , the fluid will start to transport the material, i.e.

$$P_i \geq \eta P_r \quad \text{particle movement / scour}$$

$$P_i \leq \eta P_r \quad \text{stationary particle / deposition}$$

The applied power P_r (W/m³) required per unit volume to suspend a particle with density ρ_s and settling velocity V_{ss} in a fluid with density ρ is given by

$$P_r = (\rho_s - \rho)gV_{ss}$$

3.8

A comprehensive understanding of the power approach involves an analysis of power dissipation in overcoming internal friction. The retarding effect that limits the movement of one fluid element relative to another, is the shear stress τ given by,

$$\tau = \rho g S_o (D - y)$$

2.2

Consider the movement of a small fluid element with dimensions Δx , Δy and unit width (fig 3.4).

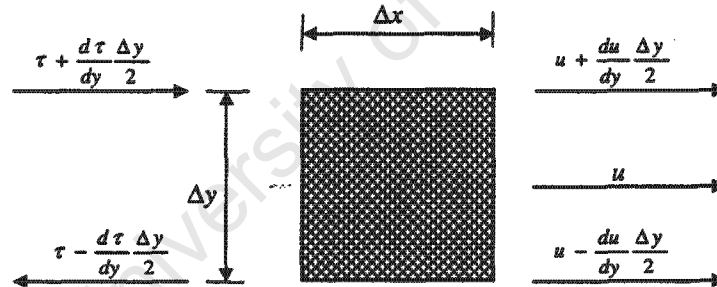


Figure 3.4 : Velocity and shear stress distribution across an element

The velocity and shear stresses vary across the element as shown. If the element translates with velocity u at an angle S_o relative to the horizontal, the power that is made available due to the steady rate of decrease in potential energy of the element is

$$\rho g S_o \Delta x \Delta y u$$

3.9

The power deficit for the element equals the work done per unit time on the upper surface of the element minus the work done per unit time on the lower surface.

$$\text{Power Deficit} = \left(\tau + \frac{d\tau}{dy} \frac{\Delta y}{2} \right) \left(u + \frac{du}{dy} \frac{\Delta y}{2} \right) \Delta x - \left(\tau - \frac{d\tau}{dy} \frac{\Delta y}{2} \right) \left(u - \frac{du}{dy} \frac{\Delta y}{2} \right) \Delta x$$

3.10

Simplifying this and dividing through by the volume of the element, gives the power deficit per unit volume as

$$\tau \frac{du}{dy} + u \frac{d\tau}{dy}$$

3.11

Differentiating and substituting equation 2.2 gives the power deficit per unit volume as

$$\tau \frac{du}{dy} - \rho g S_o u$$

3.12

The term $\tau du/dy$ represents P_t , the power applied per unit volume to deform the element or to maintain motion and the term $\rho g S_o u$ represents the amount of power made available by the element. Figure 3.5 illustrates the variations in these functions with depth.

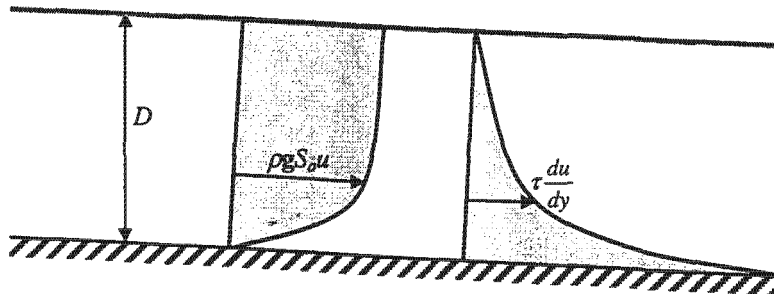


Figure 3.5 : Variation in applied power and power made available with depth
(Rooseboom 1974)

According to the power conservation principle, the areas enclosed by the graphs are equal.

There are two main conditions to consider:

(i) **Smooth Boundaries**

For laminar flow conditions in the boundary region, the velocity distribution (equation 2.15) is given by

$$\frac{u}{u_*} = \frac{u_* y}{\nu} \quad 2.15$$

Differentiating this with respect to y yields,

$$\frac{du}{dy} = \frac{u_*^2}{\nu} \quad 3.13$$

In an infinitely wide channel with steady uniform flow the shear stress u_* can be expressed as,

$$u_* = \sqrt{gDS_o} \quad 3.14$$

Substituting equation 3.14 into 3.13 and then substituting this with equation 2.2 into $P_t = \tau du/dy$ yields,

$$P_t = \frac{u_*^4 \rho}{\nu} \quad 3.15$$

for the laminar flow conditions associated with hydraulically smooth boundaries. Manipulating equations 3.15, 3.8 and 3.1 for the settling velocity of a spherical

particle, the relationship between the shear velocity and the settling velocity defined as the Movability Number (Liu 1957), can be determined in terms of a proportionality constant η (Armitage 2001).

$$\frac{u_*}{V_{ss}} = \frac{4.9\eta^{\frac{1}{2}}}{Re_*}$$
3.16

(ii) Rough Boundaries

The shear velocity given by

$$u_* = \sqrt{\frac{\tau_o}{\rho}}$$
2.14

can be substituted into the turbulent component of the shear stress equation 2.11,

$$\tau = \rho \kappa^2 y^2 \left(\frac{du}{dy} \right)^2$$
3.17

to give,

$$\frac{du}{dy} = \frac{u_*}{\kappa y}$$
3.18

Multiplying equation 3.18 by the shear stress (equation 2.14) gives the applied power P_t ,

$$P_t = \frac{\rho u_*^3}{\kappa y}$$
3.19

for the turbulent flow conditions associated with rough boundaries. The Movability Number can be found by manipulating equations 3.19, 3.8 and 3.1 (Armitage 2001).

$$\frac{u_*}{V_{ss}} = 0.69\eta^{\frac{1}{3}}$$

3.20

Equation 3.16 and 3.20 prove most useful, as many incipient motion criteria have been defined in terms of the Movability Number (Chanson 1999). Moreover, a relationship has been shown between the unit stream power approach and the shear velocity approach.

3.3.3 The Movability Number as an Incipient Motion Criterion

Rooseboom (1992) used Yang's (1973) data for incipient motion and plotted a curve relating the Movability Number to the particle Reynolds Number (fig 3.6). The grey represents the region within which most of the sediment data fell (Rooseboom 1992). The curved and straight sections represent the smooth and rough boundaries respectively. Rooseboom (1992) found that the particles would move for values of the Movability Number greater than $1.6/Re_*$ (smooth boundaries) and 0.12 (rough boundaries). The division between smooth and rough boundaries may be taken as $Re_* = 13$.

Chanson (1999) provides a table summarising the criteria found by other researches for this same relationship. Table 3.2 includes some of these results and other relationships found in the literature.

Clearly these values differ widely and are dependent on the definition of incipient motion. The overall trend of the values is summarised in table 3.3 where sliding, rolling, saltation and suspension are categorised.

Armitage (2001) related the Movability Number to the intensity of movement, i.e. the number of particle displacements during a given time interval for a particular number of surface particles under observation. It was shown that for a higher intensity of motion a greater value of the Movability Number is required. For a rough boundary,

the value of 0.17, which is associated with an intensity of approximately 1/50 000 surface particles per second, appeared to represent the beginning of movement for engineering purposes.

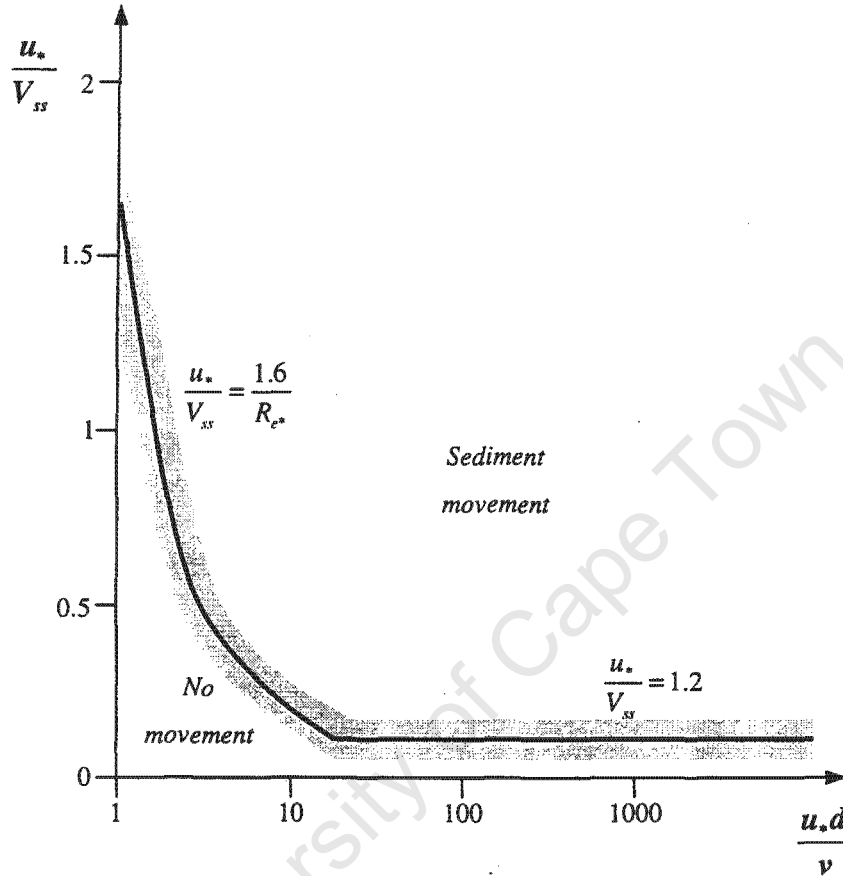


Figure 3.6 : Critical conditions for cohesionless sediment particles
(Rooseboom 1992)

Analysis of incipient motion data by Armitage (2001) suggests critical Movability Numbers of,

$$\frac{u_*}{V_{ss}} = \frac{2.0}{Re_*} \quad \text{smooth boundary}$$

3.21

$$\frac{u_*}{V_{ss}} = 0.17 \quad \text{rough boundary}$$

3.22

Reference	Criterion for suspension	Remarks
Bagnold (1966)	$\frac{u_*}{V_{ss}} > 1$	As given by Rijn (1993)
Van Rijn (1984)	$\frac{u_*}{V_{ss}} > \frac{4}{d_*}$ $\frac{u_*}{V_{ss}} > 0.4$	Experimental results for $d = d_{50}$. Suspension $1 < d_* < 10$ Suspension $d_* > 10$
Raudkivi (1998)	$0.167 < \frac{u_*}{V_{ss}} < 0.5$ $0.5 < \frac{u_*}{V_{ss}} < 1.67$ $1.67 < \frac{u_*}{V_{ss}}$	Bed load Saltation Suspended load
Julien (1995)	$\frac{u_*}{V_{ss}} > 0.2$ $\frac{u_*}{V_{ss}} > 2.5$	Turbulent flow over rough boundaries. Inception of suspension Dominant suspended load

Table 3.2 : Shear velocity: settling velocity ratio of various studies
(Chanson 1999)

Category	Movability Number	η value
(i) Sliding/rolling commences	0.12-0.17	$\frac{1}{190} \rightarrow \frac{1}{67}$
(ii) Suspension commences	0.4-0.5	$\frac{1}{2.6} \rightarrow \frac{1}{5}$
(iii) Suspension increasing in depth	0.69	1
(iv) Sediment load dominantly in suspension	2.5	50

Table 3.3 : Summary of the Movability Numbers

The associated values of η are:

$$P_t = \frac{1}{6} P_r \quad \text{smooth boundary} \quad 3.23$$

$$P_t = \frac{1}{67} P_r \quad \text{rough boundary} \quad 3.24$$

3.3.4 Rigid Body Analogy to Applied Power in Turbulent Flow

The applied stream power distribution in an infinitely wide channel (Section 3.3.2) is given by $P_t = \tau du/dy$ (Rooseboom 1992). Rooseboom (1992) also derived a formula for determining the stream power based on the fluid moving in a circular path at an instant in time. He used the Prandtl (1925) eddy model and speculated that there was an instantaneous rotation similar to that of a forced vortex around each point in the fluid. The power deficit per unit volume is then given by,

$$P_t = \left(\frac{1}{2\pi} \right)^2 \rho R^2 \left(\frac{du}{dy} \right)^3 \quad 3.25$$

where R is the radius of the eddy and du/dy is the velocity gradient across the eddy. The factor $(1/2\pi)^{0.5} \approx 0.4$, which represents the von Karman constant κ , has also been found experimentally to be approximately 0.4 for clear water.

For rigid body motion, the velocity gradient across the eddy is constant, and can be expressed in terms of the angular velocity ω_a (rad/s), as

$$\omega_a = \frac{du}{dy} = \left(\frac{u}{R} \text{ special case} \right) \quad 3.26$$

The special case occurs when the velocity is a maximum on the upper side of the eddy and the lower side is in contact with the channel bed, resulting in a velocity of zero. u (m/s) represents the average or mid-point velocity in the eddy and R (m) is the eddy radius. This concept was implemented for identifying eddies contributing to the applied stream power adjacent to the engineering structures.

3.4 Three-dimensional Unit Stream Power Distribution

The determination of the applied power P_t can be extended to three dimensions (Yang and Song 1979), by considering the governing equations of fluid for a three-dimensional fluid element. The governing equations of fluid flow and heat transfer are,

- (i) The mass of the fluid is conserved.
- (ii) The rate of change of momentum equals the sum of the forces on a fluid particle (Newton's second law)
- (iii) The rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics).

3.4.1 The Fluid as a Continuum

In considering the action of forces on a fluid, it is impossible to determine the behaviour of every molecule of fluid in a given field of flow. The problem can be simplified by considering the average effects of the molecules in a given volume. The latter implies that the fluid can be regarded as a continuum i.e. hypothetically a continuous substance. This is justified depending on the size of the volume and the number of molecules within the volume. For example, the number of molecules in a volume 10^{-19} m^3 (the size of a dust particle) would be 10^6 at room temperature and sea level. This number is sufficiently large that the average effects within the micro-

volume are almost constant. In the numerical model (Chapter 5 and 6), the fluid particles are, however, set by the mesh size.

For a continuum fluid the governing equations of fluid flow can be derived in two ways:

- (i) A finite volume fixed in space with the fluid moving through it (conservative form)
- (ii) A finite volume moving with the fluid such that the same fluid particles are always in the same control volume (non-conservative form).

A fixed element yields equations of the conservative form based on the Lagrangian system and a moving element yields equations of the non-conservative form based on the Eulerian system. For this study case (i) is employed. Both models are ultimately identical but yield different systems of equations.

Throughout this section a small fluid element is described with sides δx , δy and δz (fig 3.7).

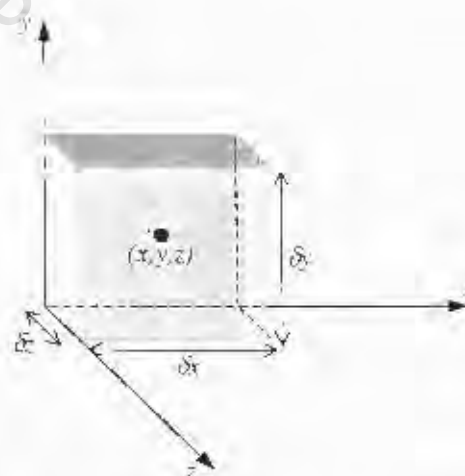


Figure 3.7 : Dimensions of a small fluid element

3.4.2 Mass Conservation

The conservation of mass implies that the rate of increase of mass in a fluid element is equal to the net rate of flow of mass into the fluid element. It can be shown (White 1991; Versteeg and Malalasekera 1995) that for an incompressible fluid i.e. density is constant,

$$\text{div} \mathbf{U} = 0$$

3.27

in other words:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

3.28

3.4.3 Substantive Derivative

The momentum and energy conservation laws make statements regarding the changes of the properties of a fluid particle. These properties are a function of position (x, y, z) and time t . Thus in an Eulerian system appropriate to fluid flow, the laws must utilise the substantive derivative for the particle. If the value of the property per unit mass is ϕ , the rate of change of ϕ for a particle with respect to time is written as,

$$\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + \frac{\partial \phi}{\partial x} \frac{dx}{dt} + \frac{\partial \phi}{\partial y} \frac{dy}{dt} + \frac{\partial \phi}{\partial z} \frac{dz}{dt}$$

3.29

where $D\phi/Dt$ is known as the 'total', 'substantive' (Versteeg and Malalasekera 1995), 'material' or 'particle' derivative (White 1991). Since a fluid particle follows the flow,

$$\frac{dx}{dt} = u \quad \frac{dy}{dt} = v \quad \frac{dz}{dt} = w$$

3.30(a-c)

and,

$$\frac{D\phi}{Dt} = \frac{\partial\phi}{\partial t} + \frac{\partial\phi}{\partial x}u + \frac{\partial\phi}{\partial y}v + \frac{\partial\phi}{\partial z}w = \frac{\partial\phi}{\partial t} + \mathbf{U} \cdot \text{grad}\phi$$

3.31

$D\phi/Dt$ is the rate of change of property ϕ per unit mass. To find the rate of change per unit volume the equation should be multiplied by the density ρ .

$$\rho \frac{D\phi}{Dt} = \rho \frac{\partial\phi}{\partial t} + \rho \mathbf{U} \cdot \text{grad}\phi$$

3.32

The first term in the equation represents the rate of increase of ϕ for a fluid particle and this is equated to the rate of increase of ϕ for a fluid element plus the net rate of flow of ϕ out of the element.

3.4.4 The Momentum Equation

Newton's second law states that the rate of change of momentum of a fluid particle equals the sum of the forces on the particle. There are two types of forces that act on a particle:

- (i) body forces - gravity, centrifugal, Coriolis and electromagnetic forces
- (ii) surface forces - pressure, viscous forces

The rates of increase of x -, y - and z -momentum per unit volume on a fluid particle are given by,

$$\rho \frac{Du}{Dt}, \quad \rho \frac{Dv}{Dt} \quad \text{and} \quad \rho \frac{Dw}{Dt} \quad \text{respectively}$$

It is common practice to include the body forces in the source terms. Consider the forces on a fluid element in the x -direction (fig 3.8). P denotes pressure forces and the viscous stresses are denoted by τ_{ij} as defined in Section 2.3.

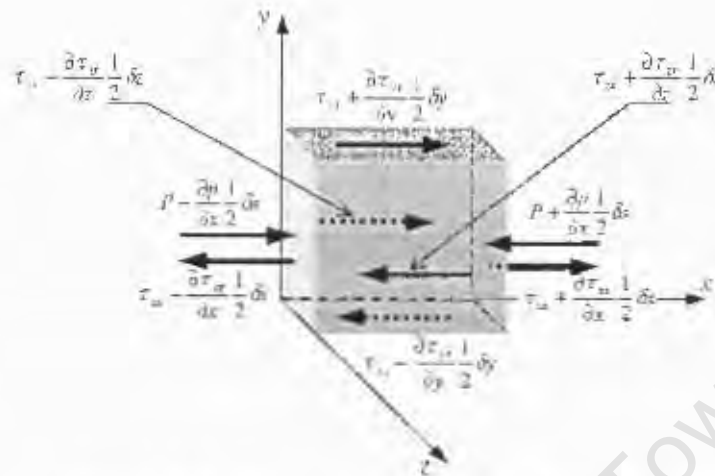


Figure 3.8 : Stress components in the x -direction

Summing these forces in the x -, y - and z -direction, with some manipulation, yields x , y and z components of the momentum equation.

$$\rho \frac{Du}{Dt} = \frac{\partial(-P + \tau_{xx})}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + S_{Mx}$$

$$\rho \frac{Dv}{Dt} = \frac{\partial(-P + \tau_{yy})}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yz}}{\partial z} + S_{My}$$

$$\rho \frac{Dw}{Dt} = \frac{\partial(-P + \tau_{zz})}{\partial z} + \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + S_{Mz}$$

3.33(a-c)

S_M is the source term containing the body force effects. In the case of open channel flow, the only body force present is gravity and hence $S_{Mz} = S_{Mx} = 0$ and $S_{My} = -\rho g$, where g is the acceleration due to gravity. P is the hydrostatic pressure, which attempts to deform the volume by compression and is hence given a negative sign.

In a Newtonian fluid the viscous stresses are proportional to the rates of deformation. In three-dimensional flow, Newton's law of viscosity involves two proportionality constants, the dynamic viscosity μ (defined in Section 2.2) and the coefficient of bulk viscosity λ , which relates the stress to the volumetric deformation. As the fluid is incompressible, the volumetric deformation is zero, and hence the nine viscous stresses are given by (Schlichting 1979),

$$\begin{aligned}\tau_{xx} &= 2\mu \frac{\partial u}{\partial x} & \tau_{yy} &= 2\mu \frac{\partial v}{\partial y} & \tau_{zz} &= 2\mu \frac{\partial w}{\partial z} \\ \tau_{xy} &= \tau_{yx} = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) & \tau_{xz} &= \tau_{zx} = \mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \\ \tau_{yz} &= \tau_{zy} = \mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)\end{aligned}\tag{3.34}$$

Substitution of 3.34 into the momentum equations 3.33a-c yields the Navier-Stokes equations, first derived independently by two scientists, Stokes (1823) and Navier (1845).

$$\rho \frac{DU}{Dt} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_{My}\tag{3.35(a-c)}$$

Substituting the body force for the source term, the equation can be written in vector form (White 1991) as,

$$\rho \frac{DU}{Dt} = -\nabla P + \mu \nabla^2 \mathbf{U} + \rho \mathbf{g}\tag{3.36}$$

i.e. the rate of change of momentum is equal to the pressure gradient plus the viscous stresses plus the body force. This equation applies to an incompressible fluid, with gravity as the only force effecting flow and a constant viscosity μ . The viscosity is

assumed constant for isothermal flow in open channels, since μ only varies as a function of temperature (Section 2.2).

3.4.5 The Energy Equation

The energy equation is derived from the first law of thermodynamics, which states that the rate of change of energy of a fluid particle is equal to the rate of heat addition to the fluid particle plus the rate of work done on the particle. The rate of increase of energy of a fluid particle per unit volume is given by,

$$\rho \frac{DE}{Dt}$$

The rate of work done on the fluid particle in the element by a surface force is equal to the product of the force and the velocity component in the direction of the force.

(a) Energy Flux to a Fluid Particle

The energy flux due to the conduction of heat is defined by the vector $\mathbf{Q}(q_x, q_y, q_z)$ where $\mathbf{Q}$ is measured in Joules. These components are depicted in fig 3.9.

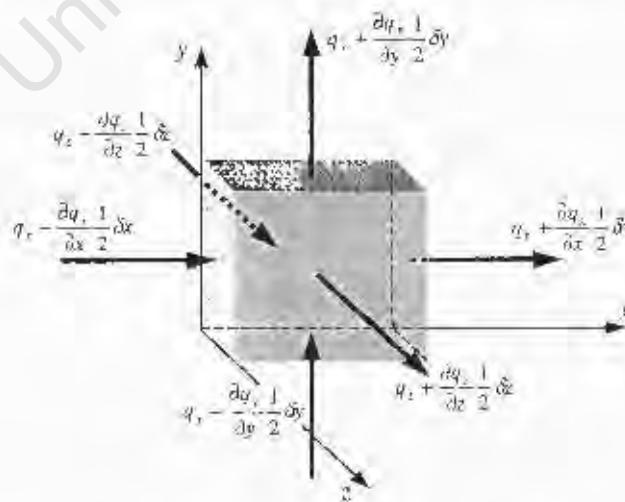


Figure 3.9 : Components of the heat flux vector

The net rate of heat addition to the fluid particle due to heat conduction across the element boundaries is given by (White 1991),

$$\frac{DQ}{Dt} = -\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} - \frac{\partial q_z}{\partial z} = -\text{div}Q \quad 3.37$$

The heat Q is positive when internal energy is transferred to a system and negative when the heat is transferred from a system. Fourier's law of heat conduction relates the heat flux to the temperature gradient by the thermal conductivity k , such that

$$Q = -k \text{ grad}T \quad 3.38$$

and hence,

$$-\text{div}Q = \text{div}(k \text{ grad}T) \quad 3.39$$

(b) Internal and Kinetic Energy Components

The energy of a fluid is defined as the sum of the internal energy i , kinetic energy $\frac{1}{2}(u^2 + v^2 + w^2)$ and the gravitational potential energy. S_E is defined as the source of energy per unit volume per unit time, which incorporates the effects of potential energy. The fluid has potential energy by virtue of position and hence the fluid flows due to gravity. The energy equation is given by,

$$\begin{aligned} \rho \frac{DE}{Dt} = & -\text{div}(PU) + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} \\ & + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zx})}{\partial z} + \text{div}(k \text{ grad}T) + S_E \end{aligned} \quad 3.40$$

where $E = i + \frac{1}{2}(u^2 + v^2 + w^2)$. The term $-\text{div}(PU)$ is zero for incompressible flow. The term $\text{div}(k\text{grad}T)$ represents the energy due to heat conduction and this is excluded as open channel flow is assumed to be an isothermal process and hence $\text{grad}T = 0$. The changes of mechanical kinetic energy can hence be expressed as (Versteeg and Malalasekera 1995),

$$\rho \frac{D\left(\frac{1}{2}(u^2 + v^2 + w^2)\right)}{Dt} = u \left(\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) + v \left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) + w \left(\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right) + \mathbf{U} \cdot \mathbf{S}_M \quad 3.41$$

after multiplying the x -, y -, and z -momentum equations (3.33a-c) with the u -, v - and w -velocity components respectively. Subtracting equation 3.41 from equation 3.40 and defining a source term $S_i = S_E - \mathbf{U} \cdot \mathbf{S}_M$ gives the internal energy equation (Versteeg and Malalasekera 1995),

$$\rho \frac{Di}{Dt} = \tau_{xx} \frac{\partial u}{\partial x} + \tau_{yx} \frac{\partial u}{\partial y} + \tau_{zx} \frac{\partial u}{\partial z} + \tau_{xy} \frac{\partial v}{\partial x} + \tau_{yy} \frac{\partial v}{\partial y} + \tau_{zy} \frac{\partial v}{\partial z} + \tau_{xz} \frac{\partial w}{\partial x} + \tau_{yz} \frac{\partial w}{\partial y} + \tau_{zz} \frac{\partial w}{\partial z} + S_i \quad 3.42$$

The term $S_i = S_E - \mathbf{U} \cdot \mathbf{S}_M = -\rho g v (-\rho g v) = 0$. From the definition of the substantive derivative (equation 3.32) equation 3.42 can be rewritten for an element as,

$$\rho \frac{\partial i}{\partial t} = \tau_{xx} \frac{\partial u}{\partial x} + \tau_{yx} \frac{\partial u}{\partial y} + \tau_{zx} \frac{\partial u}{\partial z} + \tau_{xy} \frac{\partial v}{\partial x} + \tau_{yy} \frac{\partial v}{\partial y} + \tau_{zy} \frac{\partial v}{\partial z} + \tau_{xz} \frac{\partial w}{\partial x} + \tau_{yz} \frac{\partial w}{\partial y} + \tau_{zz} \frac{\partial w}{\partial z} - \rho \mathbf{U} \cdot \text{grad} i \quad 3.43$$

3.4.6 The Dissipation Function and Stream Power

A dissipation function Φ which represents the effect on energy transfer due to viscous stresses is defined (White 1991, Versteeg and Malalasekera 1995) as,

$$\Phi = \tau_{ji} \frac{\partial u_i}{\partial x_j}$$

3.44

By substituting equations 3.34 for the shear stresses and some manipulation, Φ is given by (Yang and Song 1979, Schlichting 1979, White 1991, Versteeg and Malalasekera 1995),

$$\begin{aligned} \Phi &= \mu \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \right\} \\ &= \mu \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \frac{\partial u_i}{\partial x_j} \end{aligned}$$

3.45

The dissipation function Φ is responsible for maintaining the fluid motion. It is always positive in accordance with the second law of thermodynamics, since viscosity cannot add energy to the system (White 1991). Furthermore, the terms of equation 3.45 are all quadratic. By substituting equation 3.44 for the dissipation function into equation 3.43,

$$\rho \frac{\partial i}{\partial t} = \Phi - \rho \mathbf{U} \cdot \text{grad} i$$

3.46

where $i = cT$ and c is the specific heat of the fluid. For isothermal flow, the term $\text{grad} i = \text{grad} cT = 0$. Hence

$$\rho \frac{\partial i}{\partial t} = \rho \frac{Di}{Dt} = \Phi$$

3.47

is the rate of change of the internal energy of fluid element for incompressible, viscous, isothermal fluid flow. The dissipation function Φ is hence equivalent to the applied unit stream power P , for open channel flow (Armitage 2001), as power is defined as the rate of change of energy.

Yang (1979) stated that if the sediment concentration is not too large, the contribution of sediment load on minimum rate of energy dissipation or the stream power may be negligible. For most natural streams the change in bed profile is usually slow and the flow may be regarded as practically steady. He further found that there existed no minimum unit stream power where the Froude number was equal to or greater than one.

3.5 The Angle of Repose and Stream Power

Throughout this chapter the initiation of sediment has been considered for a particle situated on a horizontal plane i.e. the channel bed. In reality, the sediment particles are often situated on bedforms such as ripples and dunes. The particles on the positive longitudinal slope of these bedforms will be entrained more easily than the particles situated either on the adverse longitudinal slope or on a horizontal bed.

White (1940) considered a spherical particle situated on a channel bed with a particular angle of repose ϕ_s . By taking moments about the point of contact of the particle with a lower particle positioned slightly further downstream, he found a relationship between the lift and drag forces acting on the particle as well as the particle weight. This result was manipulated further by substituting the lift and drag force equations in terms of the effective velocity acting on the particle, which can be expressed in terms of the shear velocity (Armitage 2001). The Movability Number (Liu 1957) for incipient motion on horizontal channel beds is therefore adapted to include a factor $\psi = \sqrt{k_b k_r}$ such that (Armitage 2001)

$$\left. \frac{\mathbf{u}_*}{V_*} \right|_{\beta, \gamma} = \psi \left. \frac{\mathbf{u}_*}{V_*} \right|_0$$

3.48

where the suffix '0' implies a horizontal bed i.e. a Movability Number of 0.17 for saltation. β and γ are the variation of the velocity ratio with the longitudinal and transverse bed slope respectively. $\mathbf{u}_*$ is defined as the shear velocity vector which has the direction of the bed shear stress and the magnitude of the shear velocity as defined in equation (2.14). The factor ψ can be expressed as (Armitage 2001),

$$\psi = \sqrt{\cos\beta \left(1 - \frac{\tan\beta}{\tan\phi_s}\right) \cos\gamma \left(1 - \frac{\tan^2\gamma}{\tan^2\phi_s}\right)^{0.5}}$$

3.49

Equation 3.48 is essential for determining the initiation of movement for sediment particles situated on sloping channel beds.

3.6 Conclusions

This chapter has outlined the various incipient motion criteria. Attention has specifically been drawn to the unit stream power approach. Furthermore, a direct relationship between the stream power criterion and the Movability Number has been shown. The incipient motion conditions have been extended for particles situated on slopes (equation 3.48). The unit stream power approach and the Movability Number that has been adapted for topography, e.g. particles situated on scour hole slopes, are essential in predicting the scour and deposition profiles in the numerical model (Chapter 6).

Chapter 4

Scour and deposition around hydraulic structures

Prediction of scour patterns around hydraulic structures plays an important role in design. Engineering structures such as weirs, sills, piers and abutments obstruct the flow in open channels. This results in steep velocity gradients, a higher degree of turbulence and hence a stronger erosive capacity in the water, potentially resulting in scour of the channel bed.

Excessive local scour can progressively undermine the foundations of the structure. Complete protection against scour is, however, expensive. Consequently the maximum scour depth and upstream slope of the scour hole should be predicted to minimise the risk of failure (Karim and Ali 2000). This chapter outlines the types of scour that occur as well as describing the flow patterns causing scour around hydraulic structures. Recent numerical models of flow patterns and scour depth predictions are reviewed below.

4.1 Scour Types

Scour around hydraulic structures may be classified into three categories (Raudkivi 1998):

- (i) *General scour* of the stream which would occur irrespective of whether the structure was there or not, e.g. an increased flow capacity due to floods.
- (ii) *Constriction scour* arising from the narrowing/constriction of the waterway, e.g. when flows over flood plains have been intercepted by bridge causeways and channelled through the bridge waterway.
- (iii) *Local scour* resulting from the effect of the structure (e.g. piers, sills) on the local flow pattern. It results from the interaction between the high velocity

gradients and the loose bed, and the consequent modification of the flow pattern.

Hoffmans and Verheij (1997), on the other hand, classify two types of scour: general and local scour. General scour includes overall degradation as well as constriction, bend and confluence scour. Bend scour occurs due to spiral flow in the outer part of a river bend and confluence scour occurs when two branches of a river meet.

For this study, only local scour within straight channels, without tributaries, is considered, with particular emphasis on the scour due to vortex shedding around engineering structures. These vortices result in steep velocity gradients due to accelerating and decelerating flow raising the stream power (Section 3.3.4) and hence causing erosion and deposition adjacent to the structures.

Scour may be described as either clear-water or live-bed scour. Clear-water scour occurs when there is no bed material movement in the region upstream of the structure or in areas beyond the influence of the structure. The water approaching the structure is therefore 'clear' of sediment. Live-bed scour occurs under general sediment transport conditions. The water approaching the structure contains sediment, which reduces the scour/erosive capacity of the water. Figure 4.1 shows the scour depth as a function of time at a constant flow depth for the case of scour at a pier.

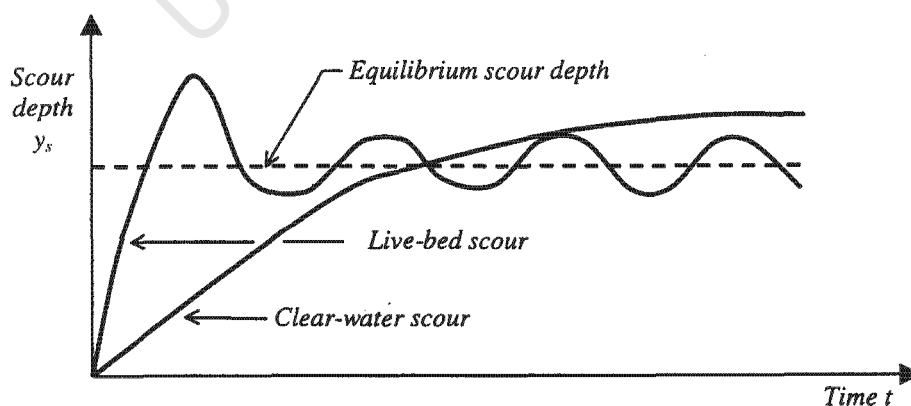


Figure 4.1 : Scour depth at a pier as a function of time (Raudkivi 1998)

In the unit stream power approach, the sediment concentration of the water is taken into account through κ (defined in Section 2.4). κ is approximately equal to 0.4 in clear water but drops for increased sediment concentrations. κ values for a variety of sediment concentrations are available in a number of texts such as Einstein and Chien (1954).

Scour prediction formulae are predominantly empirical due to the complex physics of the scour processes. Alternatively, small-scale hydraulic models are widely used.

This study tests the numerical model predictions of bead movement and scour profiles against observed laboratory measurements. The applications include the movement of beads:

- (i) over a vertical weir
- (ii) under an inverted weir or 'underflow structure'
- (iii) through a sudden contraction and over a downstream vertical weir
- (iv) through a sudden expansion and over a downstream vertical weir

and the scour profiles:

- (i) upstream of a vertical weir
- (ii) around a vertical wall abutment in rigid rectangular channel
- (iii) around a cylindrical pier centrally located in a rigid rectangular channel
- (iv) around a cylindrical pier centrally located in a self-formed channel

The following sections review the existing material on the influence of these hydraulic structures with particular emphasis on the flow patterns, the existing scour depth formulae and previous numerical predictions.

4.2 Vertical Weirs

Vertical weirs or sills can be used in rivers as energy dissipators, to maintain minimum water levels in rivers (Breusers and Raudkivi 1991) i.e. acting as low dams, to reduce the mixing of different types of water in an estuary (Hoffmans and Verheij 1997), or to measure the flow rate.

4.2.1 Flow Patterns around a Vertical Weir

The flow condition over a vertical weir, where the upstream flow is sub-critical, can be modular or drowned. If the weir is sufficiently high relative to the flow depth, it will affect the upstream water profile as the flow builds up behind the weir and passes over it at critical depth. This results in supercritical flow immediately downstream of the weir and is termed modular. If the weir is low relative to the flow depth, the upstream profile is not affected by the presence of the weir, and the flow effectively drowns it. The downstream flow has no influence on the upstream depths.

For the purpose of this study, the emphasis is on the scour/deposition profile upstream of the weir.

As the flow approaches the weir, the velocity vectors in the lower region are orientated upwards and over the weir. This causes a virtual eddy prior to the weir, with the maximum velocities passing over the weir and the minimum velocity (zero) at the bed (see fig 4.2). The water column prior to the weir is subject to an instantaneous rotation due to this velocity gradient, which is characteristic of a forced vortex (Section 3.3.4). Downstream of the structure there is a re-circulation zone, where the flow direction reverses in the vicinity of the bed (fig 4.3). This extends to the reattachment point (± 6 times the weir height downstream).

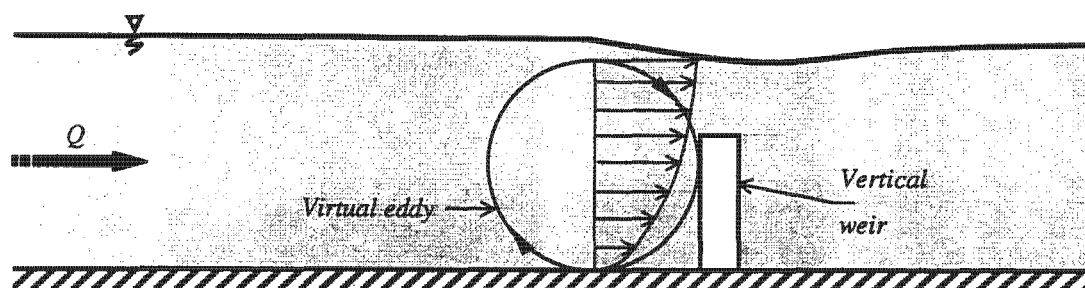


Figure 4.2 : Virtual eddy prior to a vertical weir

Above the re-circulation zone is a mixing layer. Both these regions are characterised by high turbulence energy. Beyond this is the relaxation zone, where a new boundary layer develops, leading potentially to uniform flow.

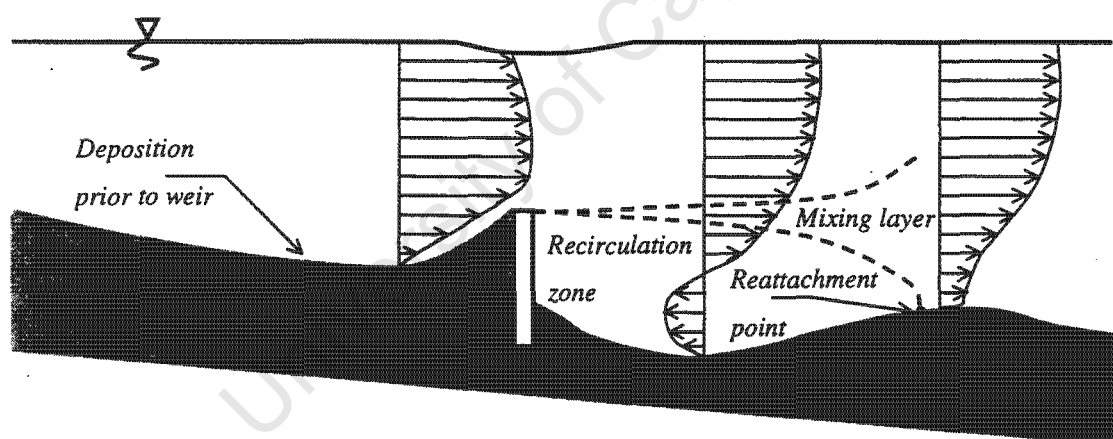


Figure 4.3 : Flow pattern around a submerged vertical weir (Hoffmans and Verheij 1997)

4.2.2 Prediction of Particle Movement and Estimation of Scour Depths

The virtual eddy upstream of the weir (fig 4.2) is the dominant factor determining whether a sediment particle will be trapped by the weir or pass over it (Compion 1997). Compion (1997) researched the transportation of litter and debris over a

folded weir and applied Rooseboom's (1992) rigid body analogy to the unit stream power theory (Section 3.3.4) for the prediction of particle movement. It was found that for $P_i > P_r$, the debris would pass over the weir and for $P_i < P_r$, the debris would be trapped.

Harnett (1998) investigated the scour profiles upstream of a weir for a variety of sediment sizes. Typically the sediment boundary was found to form a curved profile, sloping up towards the crest of the sill. In general, not much information is available about determining the curvature of the upstream section.

A number of empirically based formulae are available for determining the scour depth in the downstream section. The maximum scour depth as a function of time for clear-water scour was investigated at Delft Hydraulics (Hoffmans and Verheij 1997) based on numerous experiments. These equation parameters were later modified (Breusers 1966; Mosonyi and Schoppman 1968; Dietz 1969; van der Meulen and Vinje 1975; Zanke 1978) and in general the depth has been found to be dependent on flow velocity and bed material (Ghetti 1954; Li 1955).

4.2.3 Flow under an 'Underflow Structure'

For the purposes of this thesis, an 'underflow structure' is simply an inverted vertical weir that is suspended from the surface but does not extend to the channel bed. It has been investigated for the removal of floating debris from water surfaces (Armitage 1998). The flow pattern is approximately a mirror image of that over a weir. As the water approaches the underflow structure, the flow in the upper region is forced downwards and under the structure. This causes a virtual eddy prior to the structure, with the maximum velocities passing under the structure and the minimum velocity at the water surface.

4.3 Scour around Cylindrical Piers

4.3.1 Flow Patterns around a Cylindrical Pier

The flow pattern past a cylindrical pier positioned vertically in a uniform open channel, containing steady uniform flow, is complex, and complicated further by the scour hole development. It is characterised by the following components (Breusers and Raudkivi 1991):

- (i) downflow in front of the pier
- (ii) horse shoe vortex
- (iii) cast-off vortices and wake
- (iv) bow wave

As the undisturbed steady uniform flow approaches the pier, a bow wave forms at the free surface on the upstream side. Along the sides of the pier, the free surface is drawn down as the flow accelerates around it. Wake vortices (with a vertical axis of rotation) are formed and shed downstream of the pier as the flow about either side of the pier separates from the structure. Figure 4.4 shows the flow pattern around a cylindrical pier.

As the flow comes into contact with the front face of the pier, the velocity of the water is reduced to zero. This approach velocity distribution decreases with increased depth due to the logarithmic distribution (Section 2.6) and consequently the velocity head/stagnation pressure drops towards the bed. This sets up a pressure gradient causing a downflow on the front side of the pier. The interaction of this downward flow and the horizontal boundary layer flow close to the channel bed results in the formation of a horseshoe vortex at the base of the pier. This vortex, which has a horizontal axis of rotation, significantly effects the scouring process at a bridge pier.

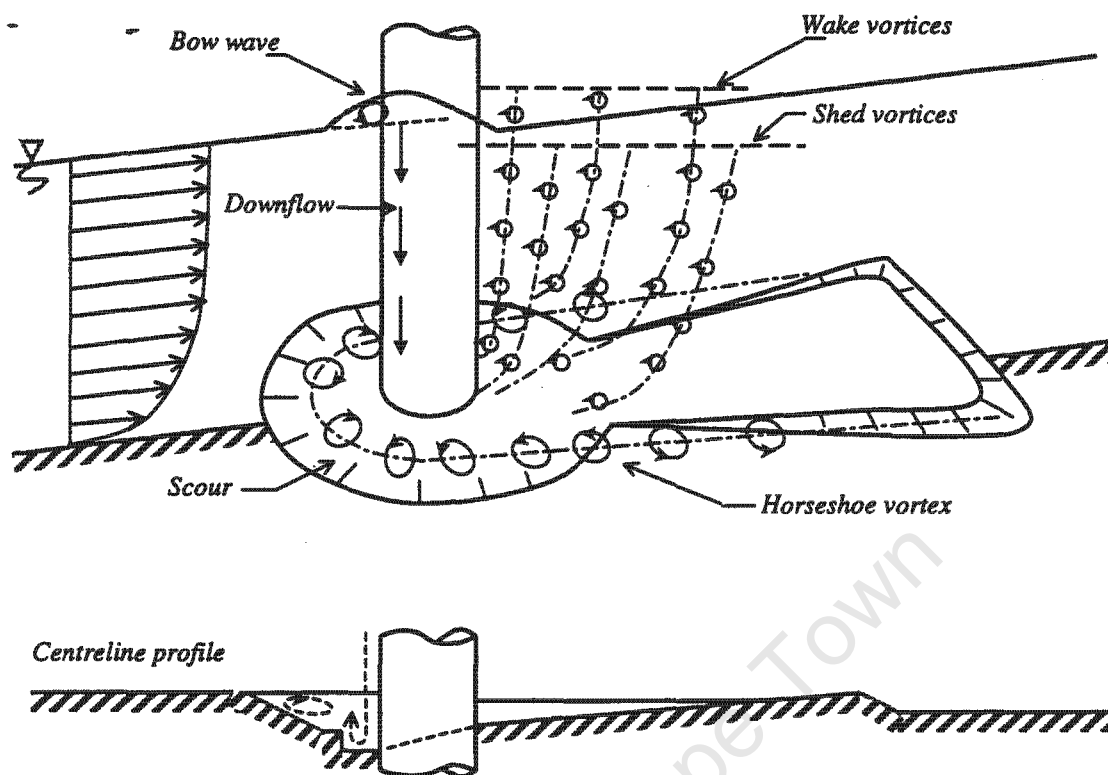


Figure 4.4 : Flow pattern at a cylindrical pier (Raudkivi 1998)

4.3.2 Estimation of Scour Depth

Local scour around bridge piers depends largely on the geometry of the pier and the waterway, together with the fluid and sediment properties (Chapter 1 and 2). The scour depth y_s can be expressed as a function of the various parameters (Raudkivi 1998),

$$y_s = f(\rho, \nu, g, d, \rho_s, D, U, b)$$

ρ = density of the fluid

ν = kinematic viscosity

g = gravitational acceleration

d = particle size

ρ_s = particle density

D = depth of flow

U = average velocity

b = pier width

Many empirically based equations for local scour have been derived (Hancu 1971; Breusers et al. 1977; Jain and Fischer 1979; Sutherland 1988; Richardson et al. 1993). Johnson (1995) provides a useful summary of the most widely used equations and the discrepancies between the scour depth values. The formulae yield widely differing estimates (up to a factor of 5 for scour depth). This is discussed in more detail by Melville (1974-5) and Hopkins et al. (1980). Neill (1973) defined the local scour depth at bridge piers in terms of the pier shape and nose curvature, while Hoffmans and Verheij (1997) found that the shape of the scour hole depends on the maximum scour depth and the side slopes of the scour hole.

Melville (1999) together with Raudkivi and Ettema (1983) derived time dependent formulae for scour depth as most calculated clear-water scour depths are overly conservative. It was found that just after the peak flow rate of a translating flood wave, clear-water scour conditions exist with sufficient time for the equilibrium clear-water scour depth to be reached. Hence, for design purposes, scour depth reduction due to live-bed conditions is negligible.

Investigation into the functional trends of scour at bridges (Raudkivi 1986) identified the main uncertainty in estimation of scour depth to be the change in pier shape due to debris (or ice rafts) caught on the pier.

Ettema et al. (1998) discussed the scale effects of pier width on local scour depth and found that the depth does not scale linearly with pier width unless there is more-or-less complete geometric similitude of pier, flow and loose bed particles – a condition that is rare. It was concluded that due to the scale effect inherent in laboratory flume experiments on local scour, and the need to better understand how changes in the local flow field affect scour, increased use must be made of computational fluid dynamics to augment laboratory experiments.

4.3.3 Numerical Models

Numerical studies have been undertaken to predict scour around bridge piers. Olsen and Melaaen (1993), used the finite volume method to solve the steady state Navier-Stokes equations and applied the $k-\epsilon$ turbulence model to solve for the Reynolds stresses. A sediment transport algorithm was incorporated but it was found that the numerical model would not handle all cases of scour around obstacles. The model would, however, calculate maximum scour depth correctly and in general, three-dimensional numerical models may be able to calculate the scour around an obstacle in a complex geometry.

Panchang and Richardson (1998) carried out a numerical study based on the parameters observed by Melville and Raudkivi (1977) and found the flow characteristics such as turbulence agreed favourably. A particle tracking technique based on the particle drag force was implemented, however the scour hole shape was required as a prerequisite. It was found that for the 'correct' drag force the particles were trapped in the swirling motion due to the presence of the scour hole. The horseshoe vortices were observed once mesh refinement had been improved. It was further found that the model boundary conditions were less sensitive to the choice of turbulent model than to the mesh refinement.

In general, the numerical techniques are most useful for situations where empirical formulae for determining scour depth are not available e.g. in tidal regimes, provided the computer resources are available.

Dey et al. (1995) derived a three-dimensional quasi-steady vortex flow field around a circular pier in a quasi-equilibrium scour hole, under a clear water regime. Although this hand model was successfully tested against laboratory data, it was stated in the concluding remarks that the solution of the Navier-Stokes equations, despite the complexity, becomes the most realistic approach to compute the three-dimensional flow field.

Takaki et al. (1996) modelled the flow field around bridge piers at a river bend to facilitate the calculation of the local scour depth. Despite assumptions such as uniform incoming inlet distributions and slip conditions on the side-walls, the prediction scour depths for flood conditions correlated well with the experimental data.

4.4 Scour around Vertical-wall Abutments

4.4.1 Abutment Types

Abutments, also termed 'groynes', 'dikes' or 'spurs', protrude into the flow from the river bank and can cause substantial local scour. Approximately one third of bridge failures in flood conditions are due to abutment scour (Raudkivi 1998). It is therefore necessary to accurately determine the scour depth and shape to facilitate a 'safe' design.

Abutments are designed in a variety of shapes: T-head, L-head, hockey, inverted hockey, semi-circular end, triangular end and so forth according to the purpose e.g. flow guide, bed protection. A more detailed discussion on shapes is contained in Hoffmans and Verheij (1997). Abutments are further classified according to the material from which they are composed, e.g. permeable, and the degree of submergence at the high water mark.

This thesis looks only at an impermeable, non-submerged, vertical-wall abutment that protrudes at 90 degrees from the riverbank.

4.4.2 Flow Patterns around a Vertical-wall Abutment

Abutments are situated at the edge of the channel. They obstruct the flow in such a way that they result in a horizontal constriction with a three-dimensional flow pattern. This thesis considers only vertical-wall, 'plate' abutments, which have a thickness t_a that is far smaller than the width b normal to the flow direction (i.e. $b/t_a > 5$).

For local scour in the region of the abutment, the interaction between the wake vortices is important. This is dependent on the ratio of the abutment height h to the breadth b . The three categories are summarised below (Hoffmans and Verheij 1997):

- (i) $b/h < 0.5$: There is a strong interaction between the upstream and downstream side of the structure. Vortices are generated intermittently from a separation point at the upstream and downstream side of the structure.
- (ii) $0.5 < b/h < 1.5$: There is a weak interaction between the vortices generated intermittently from a separation point at the upstream and downstream side of the structure.
- (iii) $b/h > 1.5$: There is no interaction between the vortices that are generated intermittently from a separation point at the upstream and downstream side of the structure (fig 4.5).

The flow approaching the abutment accelerates due to the constriction caused by the abutment. Immediately upstream of the abutment, a separation point and a small eddy exist.

As the flow comes into contact with the abutment, the velocity of the water is reduced to zero. This approach velocity distribution decreases with increased depth due to the approximate logarithmic profile (Section 2.6) and consequently the velocity head/stagnation pressure increases towards the surface. The result is a downflow on the upstream abutment face due to the pressure gradient, and a surface roller may be generated along the width of the abutment. This roller can extend around the abutment and further downstream adjacent to the abutment.

Downstream of the abutment, the main flow is separated from a large slowly rotating eddy by the vortex-street. The flow decelerates over a significant distance and uniform flow will be re-established further downstream beyond the reattachment point (fig 4.5).

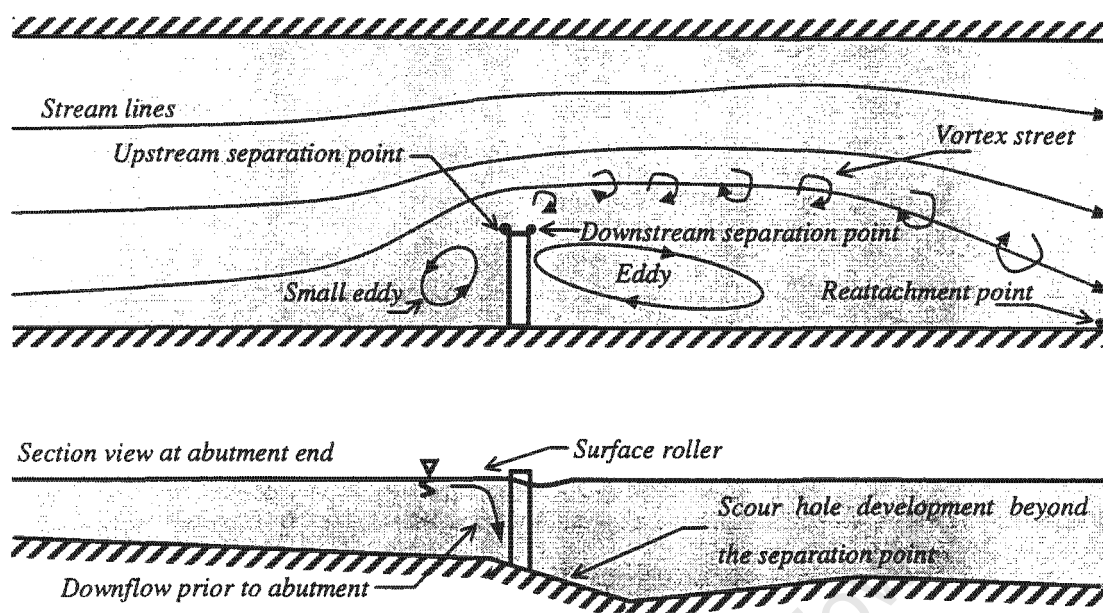


Figure 4.5 : Flow patterns around an abutment

Fiorotto (1992) observed that with development of the scour hole, the vortex in the separation zone tends to form a cone with the vertex in the deepest point of the scour near the downstream separation point. A vertical axial flow forms that can remove the sediments from the bottom of the scour hole.

4.4.3 Estimation of Scour Depth

The study of scour around abutments is limited and still lacks a reliable solution (Richardson and Richardson 1993b). The scour depth is dependent on the geometry of the abutment and the waterway, together with the fluid and sediment properties. Most existing formulae for scour depth are either empirically based (Shen et al. 1969; Froehlich 1989; Lim 1997) using rectangular laboratory flumes or alternatively based on dimensional analysis (Ahmad 1953; Lui et al. 1961; Melville 1992). Lim (1998) provides a semi-analytical approach to predicting live-bed scour at bridge abutments. A relationship between the dimensionless scour depth and the ratio of the approach velocity to the critical velocity value at initiation of motion was found by Sturm and Janua (1994). Kuhnle et al. (1999) undertook laboratory experiments around 90

degree spur dikes and found that the maximum scour depth was at the upstream separation point and that, for increased flow depths, there was a prominent secondary scour region at the downstream separation point of the dike.

Melville (1992) carried out multiple dimensional analyses followed by curve fitting to determine scour depths at vertical-wall abutments protruding at 90 degrees from the riverbank. Richardson (1992) mentioned that the formulae presented by Melville (1992) were conservative, as they do not include lateral distribution of flow, the effects of sediment characteristics or river geometric features including overbank flow.

For deep flows compared to the abutment size, Kandasamy (1989) found that the scour depth increases proportionately with abutment breadth b , but is independent of the flow depth. However, for shallow flows the scour depth increases proportionately with flow depth, but is independent of b . Ettema (1980) explained that for shallow flows the surface roller forming ahead of the abutment interferes with the scour action due to the downstream vortex, as it has an opposite sense of rotation. Mellville (1997) concluded that for deep flows the scour depth is dependent on the abutment breadth b , but this dependency reduces as the depth tends to zero.

An elongated vertical-wall abutment, i.e. an abutment with a large thickness t_a , is equivalent to one half of an elongated symmetrical contraction. Laursen (1963) provides a discussion on scour at elongated contractions and an approach to predicting the scour depths as a function of the geometry, the flow and the sediment is derived.

4.4.4 Numerical Models of Predicted Flow Fields and Scour

The use of numerical models to predict scour around abutments is still in the early phases. These models are used to either predict scour through analysis of the numerically generated flow field around an abutment or alternatively to numerically predict the actual scour depths at an abutment.

Zaghlou and Mc Corquodale (1973) developed a model capable of simulating the flow around spur-dikes by solving the Helmholtz vorticity equation and a Poisson-type equation. The model, however, was not experimentally verified.

More recently, Tingsanchali and Maheswaran (1990) modelled the flow around an abutment by solving the two-dimensional, depth averaged equations with a $k-\varepsilon$ turbulence model. The equations were solved by modifying the TEACH code developed by Gosman and Pun (1973). The results agreed with the experimental data only after a streamline curvature correction factor was applied to the $k-\varepsilon$ model.

Molls and Chaundhry (1995) developed a general mathematical model to solve unsteady, depth-averaged equations. The model uses boundary fitted co-ordinates, includes effective stresses, and is applicable to sub- and supercritical flow. The effective stresses were modelled using a constant eddy-viscosity turbulence model. A second-order accurate central differencing scheme was incorporated and the equations were solved on a non-staggered grid using an alternating-direction-implicit scheme. The velocity and shear stress values for flow around an abutment compared favourably with experimental data. The channel side-walls were modelled using a free-slip condition as the no-slip boundary condition in viscous flows can in some instances lead to inaccurate results (Yabosaki and Ponce 1981). It was found that decreased grid spacing increased the accuracy of results and the mesh size was reduced in the region immediately adjacent to the abutment. A sensitivity analysis showed that a change in turbulent viscosity did not affect the final solution, although an increase in this parameter resulted in a shorter reattachment length downstream of the abutment.

Biglari and Sturm (1998) considered flow around bridge abutments on the floodplain of a compound open channel. The model used a finite volume scheme, with a staggered grid, to solve the governing equations of flow, and a two-dimensional, depth-averaged, $k-\varepsilon$ turbulence model was incorporated. Numerical predictions of velocity in the local scour region near the abutment face, at the initiation of scour, provided scour depths that agreed favourably with those measured, for a variety of

abutment lengths and discharges. The scour depths were calculated based on a critical velocity approach.

Ouillon and Dartus (1997) investigated the three-dimensional simulation of a steady, shallow turbulent flow around a groyne in a rectangular channel. The standard $k-\varepsilon$ turbulence model was used coupled with boundary conditions for free-surface flows proposed by Rodi and Celik (1984). The method designed to track the free surface was based on a transient fraction of fluid or 'porosity field' specifying the flow domain. It was found that the computed isolines of water depths agreed well with the experimental data, however the reattachment length was underestimated. The reattachment length estimation was a marked improvement on that obtained using a rigid lid assumption. The bed shear stresses that were used to predict sediment erosion and deposition were found to agree favourably with experimental data.

4.5 Conclusions

The flow patterns and resulting scour profiles around the hydraulic structures relevant to this study have been discussed in detail. The literature has revealed that formulae based on empirical derivations and dimensional analysis cannot consistently predict the scour depth for all applications. Furthermore, no formulae exist for the calculation of the scour profile upstream of a vertical weir.

The literature also shows that there is an increasing trend towards the use of numerical models as an inexpensive alternative to laboratory models. Although no general model that is applicable for all engineering structures has yet been developed, the literature shows that in individual applications good agreement between numerical model predictions and experimental data has been found.

Chapter 5

Computational Fluid Dynamics

Computational fluid dynamics (CFD) is designed to solve numerical algorithms by computer-based simulation. The various software packages solve the governing equations of fluid dynamics and heat transfer and present the results in a graphical format. Typical applications include aerodynamics of aircraft and vehicles, hydrodynamics of ships, meteorology, air-conditioning, combustion in engines and chemical processes such as mixing, separation and polymer building. This study focuses on using CFD to solve for open channel flow characteristics and hence predict scour profiles around engineering structures.

CFX version 4.3 (1999) and version 4.4 (2001) are the commercial software packages that were used for the implementation of this study. Version 4.3 is not designed to handle open channel flow problems. Chapter 6 provides a detailed description of the adaptation of CFX-V4.3 (1999) for this study. This chapter outlines various aspects of numerical modelling and in particular the finite volume method. Version 4.4, which was released in the concluding stages of this research period, incorporates a routine for free surfaces. Version 4.4 (2001) was implemented for the final, most complex application, however, the free-surface routine was not used.

5.1 CFD Coding

The CFD programs are composed of codes that are structured around the numerical algorithms that tackle fluid flow problems. To assist the user, the package includes user interfaces to input the problem parameters and to examine the results. The three main elements are the pre-processor, the solver and the post-processor.

The pre-processor is an input interface where the problem parameters are entered. These include the dimensions for generating a mesh that simulates the physical model in a predefined co-ordinate system. Grid generation is the sub-division of the domain into a number of smaller, non-overlapping sub-domains termed 'control volumes' or

'elements'. The boundary conditions are defined at cells which coincide with or touch the-cell boundaries. The fluid properties such as density, velocity, viscosity, pressure and compressibility are defined as are the solution control parameters.

The solver reads the input files from the pre-processor, then processes the data by applying numerical solution techniques (Versteeg and Malalasekea 1995) including:

- (i) Formal integration of the governing equations of fluid flow over all the finite control volumes of the solution domain, resulting in a system of integral equations.
- (ii) Discretising these equations by the substitution of finite-difference-type approximations for the terms of the integrated equations representing flow processes such as convection, diffusion and source terms. This converts the integral equations into a system of algebraic equations.
- (iii) Solution of the non-linear algebraic equations by an iterative method.

The post-processor is a user interface in which commands can be entered to extract the relevant output information. Graphical images of the flow conditions and vector plots of variables such as pressure and velocity can be generated.

5.2 The Governing Equations of Fluid Flow

The finite volume method is distinguishable from other CFD techniques inasmuch as the governing equations of fluid flow are integrated over a control volume. This results in the exact conservation of the relevant properties, e.g. velocity and flow rate, for each finite cell. The solution to the flow parameters is defined at the nodes at the centre of each cell.

The governing equations in fluid mechanics and heat transfer are defined in Section 3.4. In Chapter 3, these equations are derived and adapted for use in the unit stream

power approach. The resulting system of equations applicable specifically to incompressible flow problems, with no heat transfer and gravity as the only body force, are summarised here:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial t} = 0 \quad (\text{mass conservation})$$

3.28

$$\rho \frac{DU}{Dt} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_{My} \quad (\text{Navier-Stokes equations})$$

3.35(a-c)

These four equations however include four unknowns; the velocities u , v and w and the pressure P .

5.3 Initial and Boundary Conditions

In order to solve the governing equations of fluid flow for a particular application, it is necessary to introduce boundary conditions.

For the purpose of modelling the flow of water around hydraulic structures, the applications fall into two categories:

- (i) Steady state, viscous flow (elliptic type equations)
- (ii) Transient, viscous flow (parabolic type equations)

For both cases it is necessary to define the boundary conditions on a solid wall, e.g. no-slip condition, and the fluid boundaries e.g. inlet velocity distribution and the continuity of stress at the outlet. For case (ii) it is also necessary to define the initial boundary conditions at time zero.

5.4 Turbulence and the Governing Equations

5.4.1 Necessity for Turbulence Modelling

The turbulence phenomenon is introduced in detail in Section 2.3. Turbulent flow is characterised by chaotic and random states of motion where the velocity and pressure change continuously with time within substantial regions of flow. The crucial difference between visualisations of laminar and turbulent flows is the appearance of eddies on a wide range of length scales in the turbulent flows. In order to identify the smallest eddies the mesh would have to be refined such that each cell size is less than $10\text{ }\mu\text{m}$ throughout the domain. Furthermore, the fastest events can take place at a frequency of 10 kHz and hence the time steps should be discretised into steps of less than $10\mu\text{s}$. Speziale (1991) states that the direct numerical simulation (DNS) of a turbulent flow pipe at a Reynolds Number of 500 000 requires a computer with a processing time that is ten million time faster than the 1991 model CRAY computer. Similarly, the modelling of an open channel of say length 1000m, width 100m and depth 10m, using DNS is not an option in the foreseeable future.

A number of turbulence models have been developed to reduce the computational load of DNS (fig 5.1).

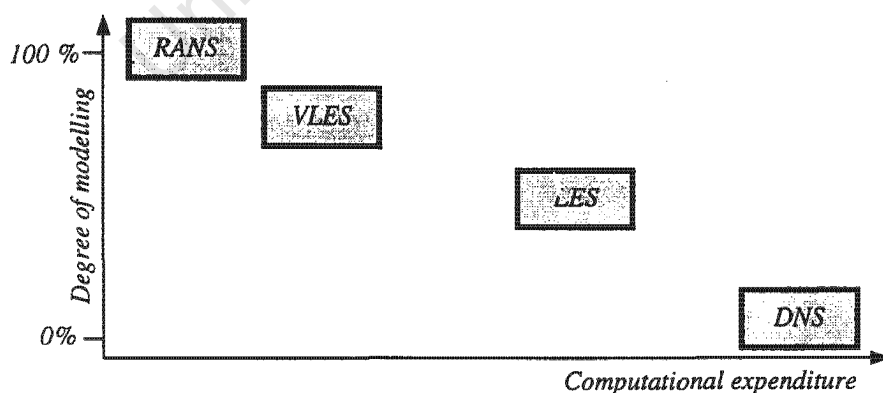


Figure 5.1 : Degree of modelling verses computational expenditure

Examples of turbulence models include the large (LES) and very large eddy simulation (VLES) models, which partially model the physical properties. These models are based on the principle that if the large eddies are accurately predicted from the mesh, the smaller eddies can be inferred. These large eddies interact with the flow domain and pass energy on to smaller eddies and thence to even smaller eddies until the viscous forces are able to dissipate the energy. This process is known as the 'energy cascade'.

The Reynolds averaged Navier-Stokes (RANS) equations are 100% based on modelling of the physical properties. These equations are introduced in more detail in the following section (Section 5.4.2) as the available models in CFX are all based on this category.

5.4.2 The Reynolds Averaged Navier-Stokes Equations

The Reynolds stress model for turbulent flow was defined in Section 2.3. Reynolds expressed the instantaneous velocity of the flow $\mathbf{U}$ in terms of the mean $\bar{\mathbf{U}}$ and fluctuating $\mathbf{U}'$ components. i.e.

$$\mathbf{U} = \bar{\mathbf{U}} + \mathbf{U}'$$

5.1

and hence the individual components can be expressed as $u = \bar{u} + u'$, $v = \bar{v} + v'$ and $w = \bar{w} + w'$. Similarly pressure can be expressed as $P = \bar{P} + P'$. By incorporating these terms in the Navier-Stokes equations (equations 3.35a-c), it is possible to re-write the velocity and pressure in terms of the average and fluctuating components. A system of equations containing the Reynolds stresses termed 'the Reynolds averaged Navier-Stokes equations (RANS)' is generated (Rodi 1984; White 1991; Versteeg and Malalasekera 1995).

$$\rho \frac{D\bar{U}_i}{Dt} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \right] - \rho \frac{\partial}{\partial x_j} \left(\overline{U'_i U'_j} \right) + S_{My}$$

5.2

These equations replace the basic form of the Navier-Stokes equations. The problem of implementing these equations is that they introduce six additional unknowns $\overline{U_i U_j}$. As mentioned in Section 2.3, Boussinesq (1877) proposed that with the onset of turbulence, the Reynolds stresses could be related to the mean rate of deformation by μ_t , a turbulent or eddy viscosity such that,

$$\tau_{ij} = -\rho \overline{U_i U_j} = (\mu_l + \mu_t) \left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]$$

2.9

This reduces the number of unknowns to one, the turbulent viscosity. There are a number of methods of determining the eddy viscosity to find closure:

- (i) The zero equation method which uses a constant eddy viscosity.
- (ii) The one equation method which uses a single equation to approximate the eddy viscosity e.g. Prandtl's (1925) mixing length hypothesis (Section 2.4) based on a length scale.
- (iii) The two equation model e.g. the k - ϵ model which introduces two equations for the transport of the kinetic energy and the rate of viscous dissipation. These are then used to approximate a length and velocity scale which are used to determine the eddy viscosity (assumes an isotropic eddy viscosity). The transport equations contain a number of empirically derived constants applicable to certain situations.
- (iv) The six equation model which introduces six new transport equations for each of the Reynolds stresses. This is termed the Reynolds stress model RSM and it solves algebraic or differential equations for $\overline{U_i U_j}$ along with a transport equation for the rate of viscous dissipation. These can then be used to approximate a length and velocity scale which are used to determine the eddy viscosity (assumes an anisotropic eddy viscosity).

Other models e.g. the low Reynolds number $k-\varepsilon$ model, contain small variations on the above mentioned models. The RNG model is derived from the re-normalisation group analysis of the Navier-Stokes equations, which involves the systematically removal of small-scale motions from the governing equations by expressing their effects in terms of larger scale motions and a modified viscosity. The Reynolds flux model can be implemented for the solution of additional scalar variables as it introduces additional equations for the scalar term i.e. $\overline{U_i \phi}$.

5.4.3 Available Models in CFX-V4.3

There are a number of available turbulence models that perform well in a variety of flow problems, however no universal model has been developed that is applicable to all situations (Chen and Wang 1996). It is therefore a fundamental design decision to select the model that is most appropriate to the particular flow situation. This selection is discussed in more detail in Section 6.6. The turbulence models available in the software package CFX V4.3 are:

- (i) $k-\varepsilon$ model
- (ii) Low Reynolds Number $k-\varepsilon$ model
- (iii) Low Reynolds Number $k-\omega$ model
- (iv) RNG model
- (v) Algebraic stress model
- (vi) Differential stress model
- (vii) Differential flux model (Reynolds flux model)

5.4.4 Applicability of models in Different Flow Situations

- (i) $k-\varepsilon$ model

The $k-\varepsilon$ model is the most widely used model. It is applicable for high Reynolds Number flows where the turbulent viscosity is isotropic. The transport equations include five empirically derived constants, which are discussed in detail by Rodi (1984) with respect to the determination and sensitivity of these constants. This

model is mostly favoured due to the short computing time (compared to the Reynolds stress models) and the simplicity of the model. It is, however, poor at predicting turbulent shear flows accurately due to the assumption of isotropic properties (Baojun et al. 1996).

The k - ε model has been found to perform well in two- (Singhal and Spalding 1975, Stephenson 1976) and three-dimensional wall boundary layers (Rastogi and Rodi 1978). Chabard et al. (1996), however, states that the k - ε model should not be applied when a precise description of the boundary layer is necessary. The k - ε model has been found to accurately predict velocity profiles around abrupt changes in geometry e.g. groynes (Durst and Rastogi 1979).

In particularly complex flows, such as those with complex geometries and complex flow structures, the results of the k - ε model may be better than those from the Reynolds stress/flux models (CFX Manual 1999).

Cokljat and Kralj (1997) found that the k - ε model is capable of predicting the main flow features, e.g. velocity and shear stress, to an acceptable level of accuracy as needed in practice. The values were sufficiently similar to those produced by the RSM and the experimental results for flows over river bedforms.

(ii) Low-Reynolds Number k - ω and k - ε models

The low Reynolds Number k - ω is based on the Wilcox (1993) turbulence model, where ω is defined as the turbulence frequency (Wilcox 1993). The low Reynolds Number k - ω and k - ε models are variations of the k - ε model for Reynolds Numbers between 5 000 and 30 000. These models do not however, introduce a wall function for the boundary layer and hence a refined mesh should be used near the boundaries.

Numerical studies on turbulent structure over square and circular strip roughness in open channel flows (Tominaga et al. 1996) revealed that the k - ε model gives excessively high predictions of turbulent shear stress and turbulent intensity for low

Reynolds Numbers. The low Reynolds Number $k-\varepsilon$ model predicted these parameters with greater accuracy.

The low Reynolds Number $k-\omega$ is a good alternative to the $k-\varepsilon$ model for determining turbulence length scale, turbulence intensities and the flow quantities. It is convenient for simulating complex flows, as it does not involve wall-damping functions, which typically require the distance from the wall (Sotiropoulos and Ventikos 1996).

(iii) RNG model

The RNG model is an alternative to the $k-\varepsilon$ model for very high Reynolds Number flows. It is derived from a re-normalisation group analysis of the Navier-Stokes equations, which involves the systematic removal of small-scale motions from the governing equations by expressing their effects in terms of larger scale motions and a modified viscosity (further details in Hoang et al. 1999). It differs from the $k-\varepsilon$ model through a modification to the transport equation for ε and it uses a different set of model constants. The advantage of this approach is that these constants are calculated explicitly.

The RNG model has been found to model curved streamlines, swirling flows and flows in non-circular pipes more accurately than the $k-\varepsilon$ model, however not as accurately as the RSM (CFX Manual 1999).

Simoes (1996) modified the RNG eddy-viscosity formula of Yakhot and Orszag (1986) for free-surface flows and found that the model predicted the bed shear stress extremely well compared with the experimental values. Furthermore, the RNG model does not require experimentally determined constants.

(iv) Reynolds Stress Models (RSMs)

The Reynolds stress models include the algebraic and differential stress models. The Reynolds flux model is an extension of the differential stress model for solving additional scalars. In these models, the eddy-viscosity hypothesis is not invoked and

the models do not make the assumption that the turbulent viscosity is isotropic. Instead, the transport equations are solved for the individual Reynolds stress and flux components. The RSM is applicable to simple flows e.g. curved streamlines, swirling flows and flows in non-circular pipes where anisotropy in the turbulence needs to be accounted for (CFX Manual 1999, Chabard et al. 1996). The RSM does include the experimentally derived constants from the transport equations for the $k-\varepsilon$ model, however the degree of influence of these is lower.

In general, the algebraic stress model takes twice the computational time of the $k-\varepsilon$ model to use and the computational expense of the differential stress model is about three times that of the $k-\varepsilon$ model. This penalty in increased CPU time is balanced by the improvement in accuracy of the solution. The RSMs should therefore only be used where there is an improvement in accuracy of the predictions (CFX Manual 1999).

The Reynolds flux model can give better predictions than the $k-\varepsilon$ model for some buoyant flows, especially stratified flows. Wallin (1996) found that the algebraic stress model results for flow in the near-wall region were similar to those obtained through DNS.

Comparison of experimental data of turbulent flows in curved channels to calculated values, showed that the near-wall second moment closure models i.e. RSMs are better than isotropic eddy-viscosity models (Chen and Wang 1996).

Perzon et al. 1996 used CFX-V4 (with some modifications to the turbulence models) to investigate complex flows over curved surfaces and found that while the $k-\varepsilon$ model predicts the pressure most accurately, the RSM provides more accurate velocity and stress distributions. Perzon et al. (1996) further stated that the use of $k-\varepsilon$ models is not advisable for flow separation and curved flows and that the RSMs are more accurate in three-dimensional flows due to the exact approximation of the source terms.

A study on numerical considerations in turbulent flow calculations (Hsich and Chang 1996) revealed that at sufficiently high Reynolds Numbers, the viscous diffusion term becomes negligible in the momentum equations. This results in numerical instability due to high values of the local grid Peclet number. As a result, additional diffusion terms based on the apparent eddy-viscosities are required in the momentum equations to ensure numerical stability.

(v) Conclusions

This extensive review and comparison of the model performances in various applications outlines the difficulty in selecting the most appropriate model. The model selection was therefore based partially on modelling situations with known results and partially on the advice gleaned from this review. This is discussed in more detail in Section 6.6.

5.5 Differencing Schemes

Once the governing equations have been adapted to incorporate the appropriate turbulence model, these integral equations are discretised. This involves the substitution of a variety of finite-difference-type approximations for the terms in the integral equations representing flow processes such as convection, diffusion and source terms (Versteeg and Malalasekera 1995). The result is that the integral equations are converted into a system of non-linear algebraic equations.

Differencing schemes are assessed in terms of their conservativeness, boundedness and transportiveness. These parameters are discussed in detail in texts such as Versteeg and Malalasekera (1995) or Ferziger and Milouan (1996). The transportiveness is measured in terms of the Peclet number, Pe , defined as

$$Pe = \frac{\rho\mu}{\Gamma\Delta x}$$

5.3

where ρ (kg/m³) is the fluid density, μ (Ns/m²) is the fluid dynamic viscosity, Γ is the co-efficient of diffusion and Δx (m) is the distance between the nodes. The CENTRAL differencing method is only bounded for values of the Peclet number less than two and it does not recognise flow direction, hence it is not transportive at high values of Pe . The UPWIND scheme is bounded for values of the Peclet number greater than two.

The differencing schemes available in CFX-V4.3 are:

- | | | |
|--------|---------------------|---|
| (i) | UPWIND | first order accurate |
| (ii) | Higher Order UPWIND | second order accurate |
| (iii) | CENTRAL | second order accurate |
| (iv) | HYBRID | first order accurate |
| (v) | QUICK | third order accurate – advection
second order accurate - diffusion |
| (vi) | CCCT | (same as QUICK) |
| (vii) | CONDIF | second order accurate |
| (viii) | MUSCL schemes | third order accurate |

The UPWIND scheme uses the concept that the value of a variable is more dependent on the value of the previous node in the direction of flow.

The higher order UPWIND scheme is an upwind scheme made second order accurate by extrapolating to two upwind points.

The CENTRAL differencing scheme is based on simple linear interpolation.

The HYBRID scheme is a combination of the UPWIND and CENTRAL methods. For $Pe < 2$, the central differencing method is used and for $Pe > 2$, the upwind scheme is used.

The quadratic upwind differencing scheme (QUICK) takes into account the lower order terms of the Taylor series. A quadratic function is used to determine the values at the cell walls.

The CCCT scheme is a modification of the QUICK scheme that is designed to correct for non-physical solutions in higher order schemes. E.g. negative turbulent kinetic energy.

The CONDIF scheme is a modification to the CENTRAL difference scheme to ensure a diagonally dominant matrix.

MUSCL or TVD schemes are modifications to the higher order upwind scheme with flux limiters to ensure boundedness of the solution.

5.6 Pressure Correction

Once the governing equations have been discretised to form a system of algebraic equations, it becomes necessary to apply a pressure correction. This arises from the non-linearity of the momentum equations and the velocity-pressure coupling. The momentum and continuity equations are intricately coupled as every velocity component occurs in each equation. The flow field that is solved for may incorrectly satisfy these equations since a highly non-uniform pressure field may appear to be uniform.

The solution is to introduce a staggered grid for the velocity components while the other flow variables are solved for and stored at the original nodal points (Harlow et al. 1965). A number of algorithms have been developed for this purpose (details in Ferziger and Milouan 1996; Versteeg and Malalasekera 1995). Those available in CFX-V4.3 (1999) are:

- (i) SIMPLE
- (ii) SIMPLEC

- (iii) PISO
- (iv) - Non-iterative PISO

5.7 Solution Control

Once the pressure correction has been applied, the system of non-linear algebraic equations can be solved. Due to the non-linearity of these equations, there is an outer iteration and an inner iteration.

The inner iteration is solved for by means of a linear equation solver, which uses an iterative solution method. An exact solution is not required because this is just one step of the non-linear outer iteration. The iterative solvers available in CFX-V4.3 are:

(i)	Line relaxation	LINE SOLVER
(ii)	Preconditioned conjugate gradients	ICCG
(iii)	Full field Stone's method	STONE
(iv)	Block Stone's method	BLOCK STONE
(v)	Algebraic multi-grid	AMG
(vi)	General version of the algebraic multi-grid	GENERAL AMG

These solvers ultimately lead to the same solution, although certain solvers perform better in certain instances.

The outer or 'global' iteration is then solved. The non-linearity of these original equations is simulated by re-forming the coefficients of the discrete equations, using the most recently calculated values of these variables before each outer iteration.

The stopping criteria for the outer iteration may be defined in terms of the number of iterations, the number of time steps (transient problem), the mass source tolerance on a residual or the nature of the residual plot. The most commonly used method throughout this study is the mass source tolerance, which is given by,

- (i) Explicit scheme : at time t
- (ii) Crank-Nicolson scheme : equally dependent on time t and time $t + \Delta t$
- (iii) Implicit scheme : time $t + \Delta t$

In CFX-V4.3 (1999), fixed or adaptive time stepping can be used, or a combination of the two. Fixed time stepping involves defining a fixed time interval for each step and adaptive time stepping provides an automatic control of the time step so that convergence is achieved with maximum efficiency. This avoids the need for human intervention in long transient runs.

5.9 Conclusions

The numerical methods and their relevance to the open channel flow application have been outlined, with specific reference to the software package CFX-V4.3 (1999). This section is vital for interpreting Chapter 6, where the adaptation of CFX-V4.3 (1999) for the purpose of modelling open channel flow around engineering structures is described. The available options in CFX have been presented and Chapter 6 explains the reasoning for each option that was selected.

Chapter 6

Adaptation of CFX-V4.3 (1999) for Open Channel Flows

CFX version 4.3 (1999) is the computational fluid dynamics software that was the fundamental modeling tool throughout the implementation of this study. This software is designed to handle a variety of fluid problems ranging from multiphase flows in chemical reactors to the drag force around spherical balls suspended in air. It does not, however, have a routine specifically designed to model free-surface flow.

This chapter gives a detailed description of how CFX-V4.3 (1999) has been adapted for the purposes of modeling open channel flow. The creation of a representative geometry of the flow domain in the vicinity of the engineering structure is discussed, together with the degree of mesh refinement. Published data of shear stress and turbulent viscosity distributions are employed in calibrating the program with respect to the turbulence model selection. The benchmark problem of a lid-driven cavity is utilised in selecting the differencing scheme and turbulence criteria. A final methodology is outlined that includes the selection of the pressure correction, the solvers and the additional subroutines that were written into the software code.

CFX-V4.4 (2001), which includes a free surface routine, was released in the concluding stages of this research period. This version was used for the final, most complex application although the free surface algorithm was not implemented.

6.1 Mesh generation

6.1.1 Creating the Geometry

The first step in solving a fluid problem involves creating a representative geometry in CFX Build. The geometries that were created to represent the engineering structures were predominantly rectangular blocks that represented the channel, weirs, suspended sills, contractions, expansions or abutments. In creating the piers, half cylinders were formed using arcs, whilst for the purpose of modeling dunes, profile

co-ordinates were entered and linked to form solids. For uneven surfaces, where no particular function was available to describe the surface, a number of points were entered and a spline was used to create a best-fit or representative surface. For creating sediment contours, the co-ordinates at regular intervals along each contour were entered. These were then linked up as a chain. Each contour height chain was then connected to the previous height by means of a surface. This final uneven surface representing the sediment topography was then connected to the water surface to create a solid.

6.1.2 Refining the Mesh at the Bed

Once the geometry was created, the domain was divided into finite volumes. An increased number of cells results in increased CPU time, and in some instances, it results in data files that are too large to manipulate in the available post-processors. Consequently, CFX-V4.3 provides the option of biasing the mesh using a one or two-way bias and of specifying the exact dimensions of the end cells.

The area of interest was the bed region and in order to determine the applied power as a function of the velocity gradient, the mesh in the region of the bed had to be finer than the thickness of the laminar sub-layer. In the laminar sub-layer, the velocity gradient is constant (see Section 2.6). To determine the size of this cell, the value of $y^+ \approx 11.8$ at the intersection of the turbulent layer and the viscous sub-layer was estimated from

$$y^+ = \frac{u_* y}{\nu} \approx 11.8$$

2.18

The shear velocity was determined from

$$u_* = \sqrt{\frac{\tau_o}{\rho}}$$

2.14

Initially the bed shear stress is unknown. The approach channel was therefore modeled initially with a coarse mesh to determine the largest shear stress at the channel bed. This was then used to calculate the shear velocity (equation 2.14) and the value of y (equation 2.18). The smallest cell dimension was made slightly smaller than this.

The command file gives the option of defining the sub-layer thickness and log-layer constant, which are known entities. The program, however, is designed to approximate these correctly without further user input (Fletcher 2000).

For hydraulically rough conditions, the mesh should also be finer than the roughness height k_s of a sediment particle (fig 6.1).

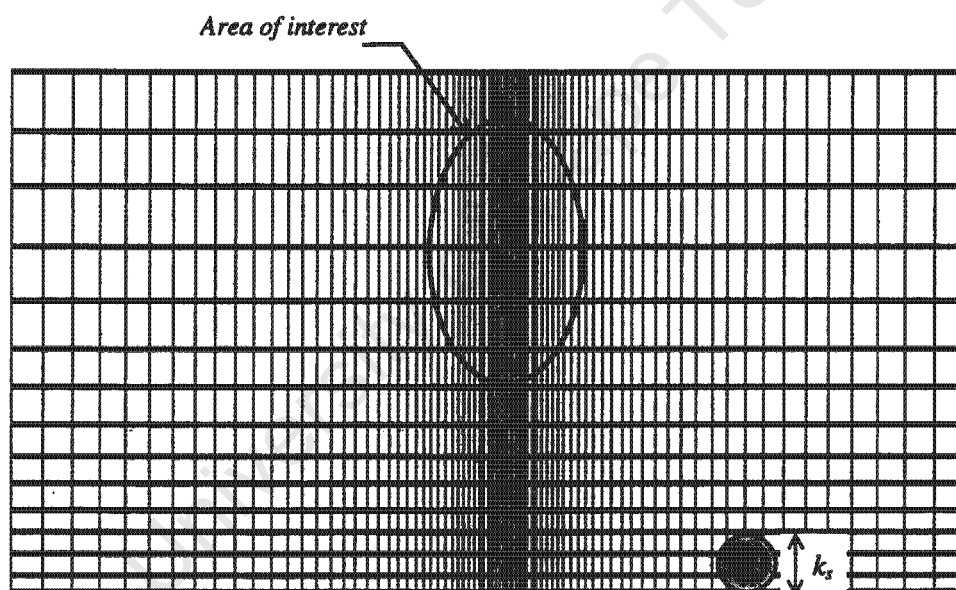


Figure 6.1 : Mesh refinement around a particle and in an area of interest

6.1.3 Refining the Mesh in Area of Interest

Although important, the bed was not the only region of interest. Eddies are formed in the vicinity and downstream of the structures (Chapter 4) and hence the mesh was made smaller in these regions (fig 6.1).

6.2 Programming Response

CFXV-4.3 is programmed in Fortran 77. In the course of the implementation, additional subroutines were written to determine the power dissipation

$$\Phi = \tau_{ji} \frac{\partial u_i}{\partial x_j}$$

3.44

which may be written as,

$$\begin{aligned} \Phi &= \mu \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \right\} \\ &= \mu \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \frac{\partial u_i}{\partial x_j} \end{aligned}$$

3.45

and the Movability Number. The velocity gradients are required in the dissipation function Φ (equation 4.35).

USRTRN is an available user routine in CFX for calculating additional quantities from the basic solution variables. A built-in routine for determining velocity gradients, GRADV, is also available in CFX. These gradients do not form part of the general output and thus the first routine that was written, USRTRN, was designed to call GRADV and specify 'user scalars' equal to each gradient term. To do this a workspace was created to store these variables using the subroutine 'SETWRK'. They were later deleted using another routine, termed 'DELWRK'. These subroutines were all called in the user routine USRTRN. The user can hence specify the sections of the domain in which the new quantities should be calculated using utility routine IPREC or IPALL by predefining user two- or three-dimensional patches in the CFX build section i.e. when creating the geometry. For most cases the quantities were determined throughout the domain. Figure 6.2 describes the subroutine hierarchy (see routine in Appendix, A1).

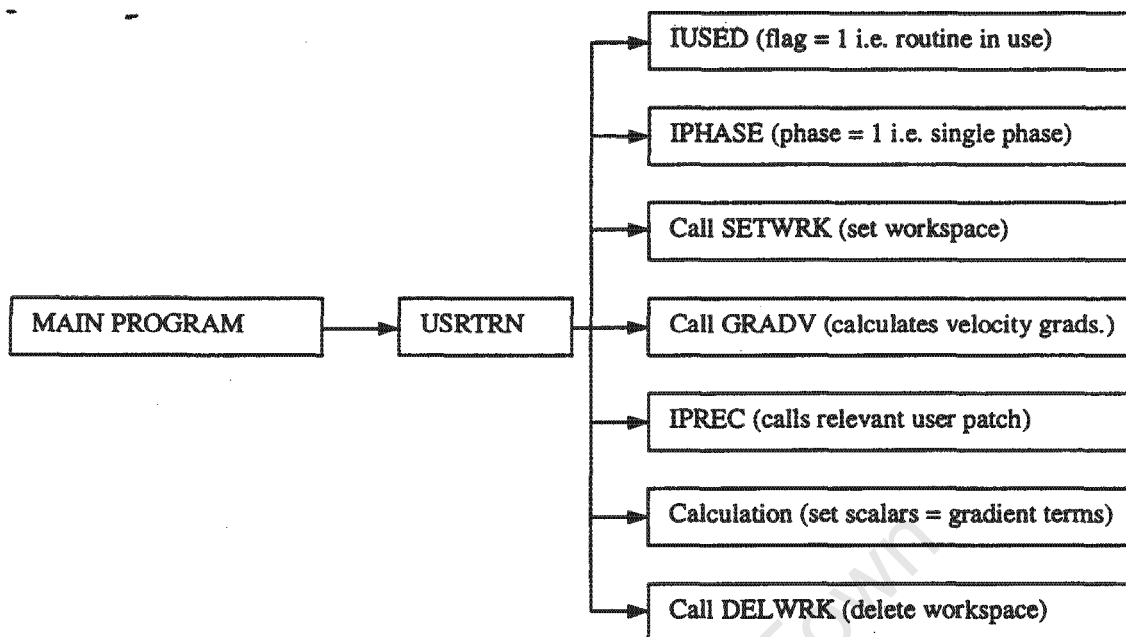


Figure 6.2 : Flowchart of subroutine for velocity gradients

This routine was tested by solving for the velocity and velocity gradient terms for the benchmark problem of the flow between two parallel plates. The top and bottom plates were set as moving walls at 10 m/s and 5 m/s respectively. The results showed the correct linear velocity distribution between the plates, however the velocity gradients in the boundary region were underestimated (fig 6.3). It was found that all user scalars should be specified as USRDCC followed by the scalar name in the command file. This tags the scalar as one which does not require solution in the transport equations. These variables are then treated as places to store arbitrary information. This produced the correct velocity gradient at the boundary.

Once accurate velocity gradients were obtained, this routine was used for programming the power function (equation 3.45) (Appendix, A2). The shear stresses are required for equation 3.44 as well as in determining the shear velocity for the Movability Number using equation 2.14. These shear stresses could be obtained from the Reynolds stress array, which is accessible in user routine USRTRN (Appendix, A2). This routine was used for any further functions that did not require the transport equations to be solved.

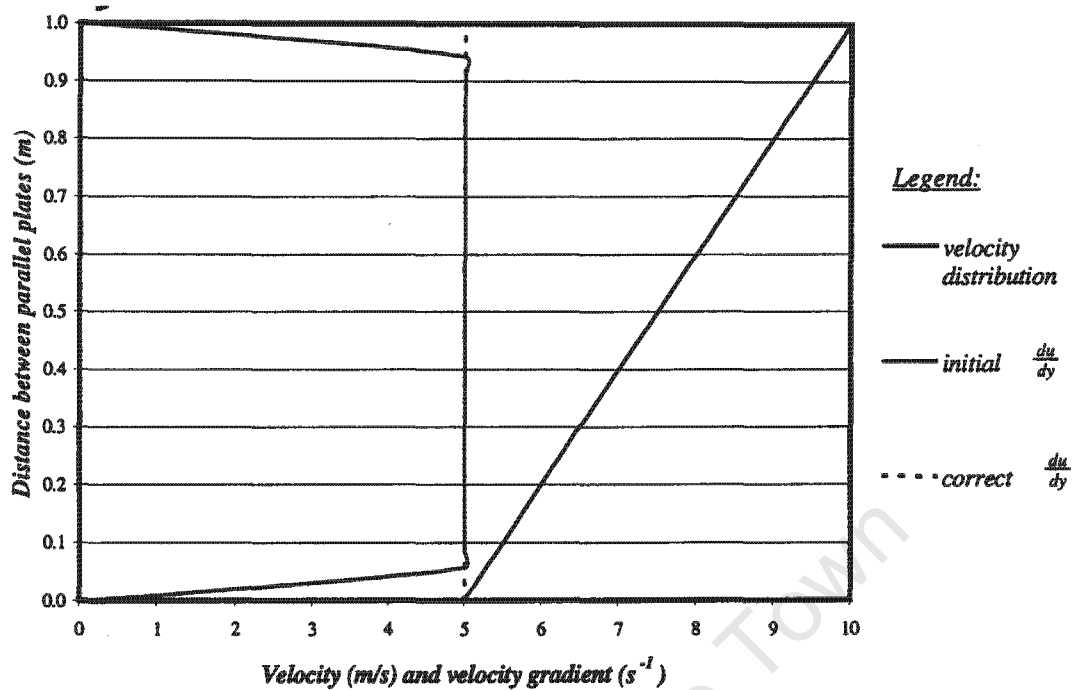


Figure 6.3 : Velocity and velocity gradient distribution between parallel plates

6.3 Boundary Conditions

6.3.1 Inlet Velocity Profile

Steady uniform flow in a rectangular open channel has a fully developed boundary layer (Section 2.6). The upstream boundary condition, or inlet, was assumed to be that of steady uniform flow. To reduce the length of the numerical model, a logarithmic velocity function characteristic of the turbulent boundary layer, was specified for the inlet velocity distribution. The logarithmic velocity profile can be expressed as the point velocity at a particular depth (Rooseboom 1992),

$$\frac{u}{u_*} = \frac{1}{\kappa} \ln \left(\frac{y}{y_o} \right)$$

2.25

or alternatively in terms of the average velocity (Rooseboom 1992),

$$\frac{\bar{u}}{u_*} = \frac{1}{\kappa} \ln \left(\frac{D}{ey_o} \right)$$

2.27

where D is the flow depth. See Section 2.8 for more details. The velocity distribution in terms of the average velocity can thus be found by substituting equation 2.25 into 2.27 giving,

$$u(y) = \frac{\ln \left(\frac{y}{y_o} \right)}{\ln \left(\frac{D}{ey_o} \right)} \bar{u}$$

6.1

It is possible to adapt this distribution for a rectangular channel. The velocity distribution in the zy -plane for an axis placed at the channel centre is given by,

$$u(y, z) = \frac{\ln \left(\frac{y}{y_o} \right) \ln \left(\frac{\frac{B}{2} - |z|}{z_o} \right)}{\ln \left(\frac{D}{ey_o} \right) \ln \left(\frac{\frac{B}{2}}{ez_o} \right)} \bar{u}$$

6.2

where B is the channel width and z_o is the ordinate on the channel wall at which the velocity is mathematically equal to zero. z_o can be determined similarly to y_o from equation 2.26 (Section 2.8) by determining the side wall roughness coefficient. It can be shown that integration of equation 6.2 over the channel area gives the total flow rate Q ,

$$Q = \int_{z_0}^B \int_{y_0}^D u(y, z) dy dz = \bar{u}BD = \bar{u}A$$

6.3

for the channel.

Equation 6.1 and 6.2 were used as the inlet velocity function for two- and three-dimensional analyses respectively. The user routine USRBCS was used to incorporate this function. USRBCS is a routine enabling the user to specify more complicated boundary conditions such as profiles of variables across inlets, pressure boundaries and mass flow boundaries. Within the user routine, utility routine GETVAR is used to access the velocity array and IPREC is called to indicate the specific block or patch where the addresses of cells centers or vertices are required e.g. the patch name inlet. The IUBCSF flag is set to determine the nature of the boundary condition variation with time and iteration. Figure 6.4 describes the subroutine in more detail (see routine in Appendix, A3).

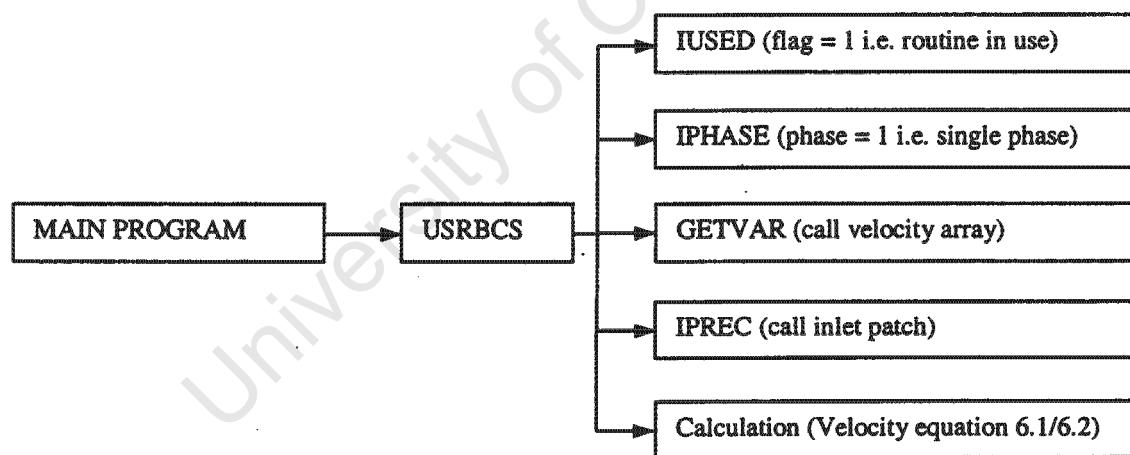


Figure 6.4 : The inlet velocity distribution subroutine USRBCS

Once the program has been run, the output file indicates the total incoming flux, and this provides an additional check that integrating the velocity profile yields the correct flow rate. Fujisaki et al. (1999) provided useful experimental data for the velocity distribution in a three-dimensional channel.

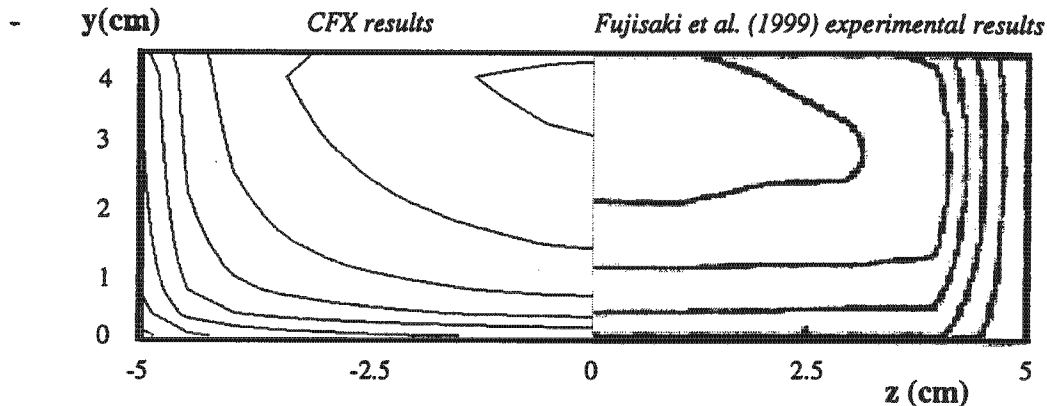


Figure 6.5 : Comparison of CFX velocity profile to experimental data

The distribution computed by CFX was reasonably similar to that of Fujisaki et al. (1999) at a section some distance downstream of the programmed inlet distribution (fig 6.5).

6.3.2 Outlet

This downstream boundary condition, where the flow is anticipated to leave the representative channel geometry, was specified as the outlet patch. This condition implies that:

- (i) The gradients of all the transported materials are specified rather than their values. All transported quantities are given zero normal gradient, with the exception of velocity, which is given a constant normal velocity gradient.
- (ii) Fully developed flow occurs at the outlet i.e. the outlet is far downstream.
- (iii) The pressure is extrapolated from upstream conditions.
- (iv) Mass continuity is satisfied.

6.3.3 Symmetry

The symmetry condition is used when modelling two-dimensional flow cases e.g. a section of an infinitely wide channel. It is also implemented when modelling a symmetrical structure e.g. a pier, thus facilitating reduction in the number of cells used.

6.3.4 Walls

The default value for any unspecified boundary is 'wall'. These walls can be renamed individually where it is necessary to describe additional wall information. Boundaries of the flow domain representing the channel bed were set as walls, and were given the no-slip condition typically used for viscous fluids. Logarithmic wall functions were also specified.

As the flow conditions for sediment transport were being modeled, it was necessary to specify the wall roughness such that the program calculated the correct logarithmic distribution (Section 2.7). This was implemented by incorporating a roughness routine, termed USRWTM, which allows the user to enter a roughness height and a roughness factor (Appendix, A4). Based on these the non-dimensionalisation of the distance from the wall in the 'log-layer' formula is modified along with the log-layer coefficient (Section 2.6). The roughness height used was that of the sediment dimension d_{50} (Chapter 3). A value of 0.3 for the roughness factor indicates fully rough conditions and a value greater than 0.3 would have smooth conditions. A sensitivity analysis on this parameter showed that the results did not change for values of the roughness factor up to 0.9.

6.3.5 Free Surface

There is no free-surface algorithm in CFX-V4.3 (1999). The free surface can, however, be approximated in a number of ways e.g. symmetry condition, a rigid-lid assumption (Ouillon and Dartus 1997; Sotiropoulos and Ventikos 2000) or depth averaged equations (Molls and Chaudry 1995; Biglari and Sturm 1998). CFX-V4.4

(2001), which includes a free surface routine, was released in the concluding stages of this research period, but the free surface algorithm was not investigated.

The options that were attempted for CFX-V4.3 (1999) included a symmetry condition, a rigid-lid with an air interface and a moving wall.

(i) The Symmetry Condition

The first option was to model the surface patch as a boundary parallel to the bedslope at the experimentally measured flow depth with a symmetry condition. This symmetry patch was not a good approximation for the water surface, as the turbulent viscosity distribution (Section 2.3) was incorrectly predicted (fig 6.6).

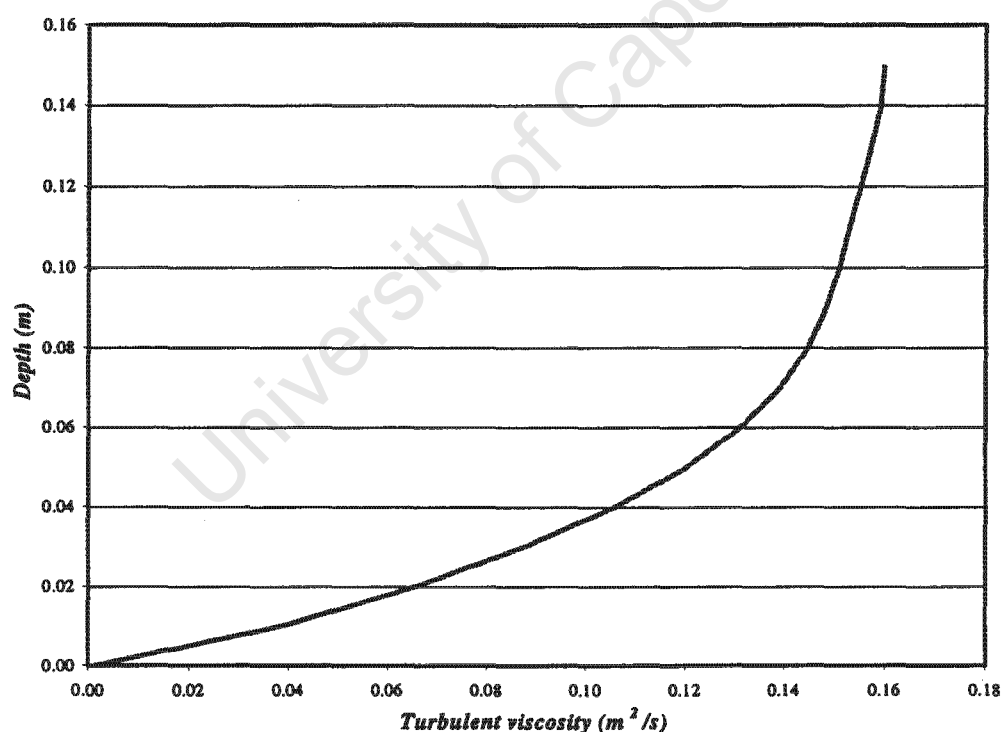


Figure 6.6 : Turbulent viscosity variation with depth

The viscosity distribution increased away from the channel bed as expected, however, it continued to increase towards the water surface in a similar manner to the distribution observed in pipe flow (Schlichting 1979). The turbulent viscosity is a primary input in the power equation (equation 4.49) and therefore this option was not suitable.

(ii) The Rigid Lid

The second option was to model the channel with an air-pocket above the water surface and a rigid lid above this, along which the pressure could be monitored to ensure atmospheric conditions were present. This situation was modelled as a transient problem as the water surface had to reach an equilibrium profile around the hydraulic structure. The problem was further complicated by the presence of two phases, air and water. Figure 6.7 shows the final boundary conditions that were chosen.

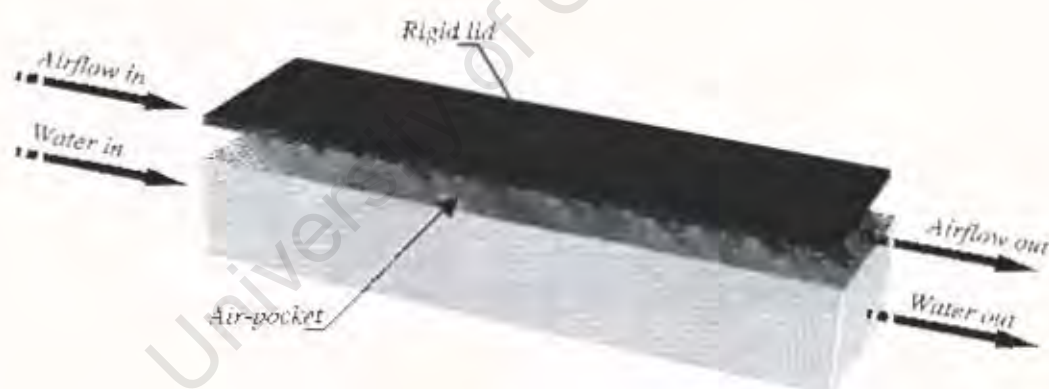


Figure 6.7 : Rigid lid model

The air-flow was modelled as uniform with the velocity of the water surface. The initial model conditions were set in the routine USRINT. The initial condition was for the volume fraction of water in the bottom section to be set as one and moving with a logarithmic velocity profile. The bottom and top sections were specified by means of a user three-dimensional patch. The volume fraction of air in the top section was set as one and moving at a velocity equal to the surface velocity of the water. The ambient conditions at the inlet and outlet boundaries were also set in terms of the

volume fractions. The problem was run with fixed time stepping initially, followed by adaptive time stepping.

The final water surface was similar to the profiles measured in the laboratory for the case of two-dimensional flow in a rectangular channel. However, as soon as an obstacle was introduced e.g. a weir, convergence difficulties were encountered. The problem was run with different combinations of solver, differencing scheme, under-relaxation factors, pressure correction and time-stepping options. The solution, however, repeatedly diverged.

(iii) The Moving Wall

The final option was to model the surface as a moving wall. The flow conditions that were being modelled were characterised by approximately flat surfaces, with sub-critical flow and moderate energy losses. The CFX turbulence models are all functions of some length scale and a velocity scale, where the length scale can be interpreted as some measure of the distance from the nearest wall (Prandtl 1925; Rooseboom 1992). As the turbulence model determines the turbulent viscosity, modelling the surface as a moving wall would correct the profile from that obtained in the symmetrical case.

The wall was modelled with the surface velocity of the water to avoid shearing of the layers against this wall (fig 6.8).



Figure 6.8 : Moving wall model

and dissipation length scale. This, however, results in a uniform distribution of these quantities at the inlet whereas in fully developed flow the distribution of k increases linearly towards the channel bed and the ϵ distribution is small at the surface increasing at an increasing rate towards the channel bed. As the shear stress is dependent on these quantities, the correct distributions had to be programmed in at the inlet patch.

Each model was initially run over a length of approximately 60 times the channel width, with a coarse mesh. The k and ϵ distribution at the outlet were then plotted in Microsoft Excel and a best-fit polynomial was used to describe these profiles. These distributions were then programmed in at the inlet patch of the actual model with the refined mesh (fig 6.11). In the three-dimensional case the distribution of these quantities at mid-height was plotted along with the vertical distribution to generate an equation describing the surface distribution at the inlet. These parameters were programmed in USRBCS along with the velocity distribution (Appendix, A6). This resulted in a linear shear stress distribution at the inlet patch.

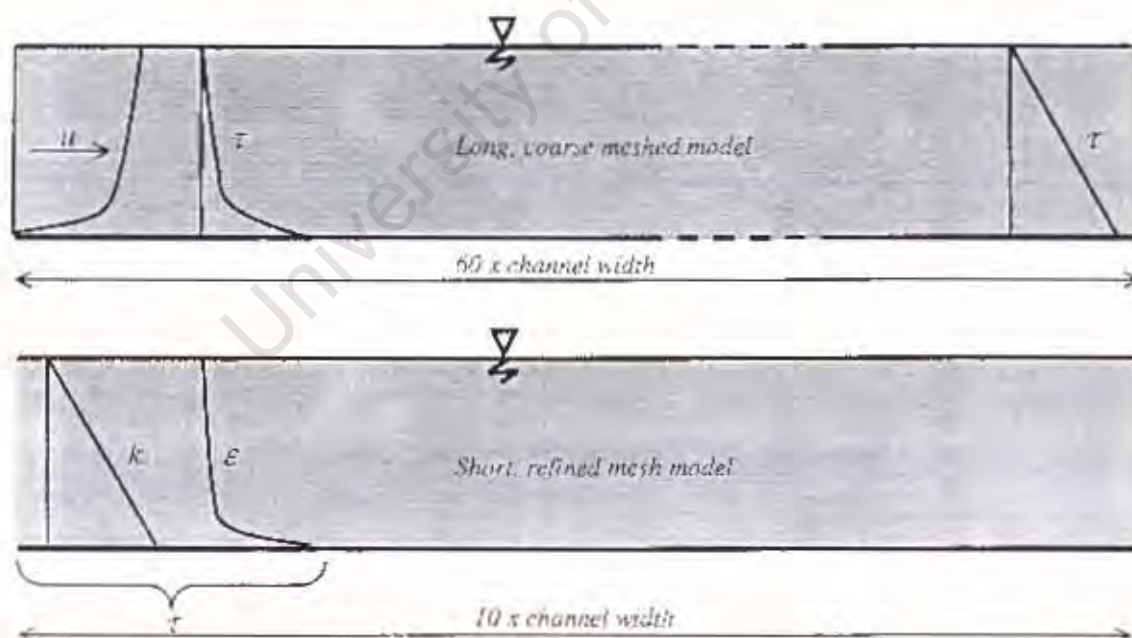


Figure 6.11 : Revised shear stress profile – setting k and ϵ

The velocities were programmed in the routine USRBCS by selecting the index for the horizontal velocity equation, and setting the parameters A and B for the velocity and shear stress wall functions and hence the variable C as the desired function. The velocity equations 6.1 & 6.2 (Section 6.3.1) were implemented however the y -variable was fixed at the flow depth. The top wall cell centres were selected by means of a user two-dimensional patch and utility routine IPREC was used to identify this patch. Figure 6.9 describes the USRBCS routine (see routine in Appendix, A5).

This model proved the most successful of the three models and it gave an improved turbulent viscosity distribution (Section 2.3) compared to the symmetry surface patch condition (fig 6.10).

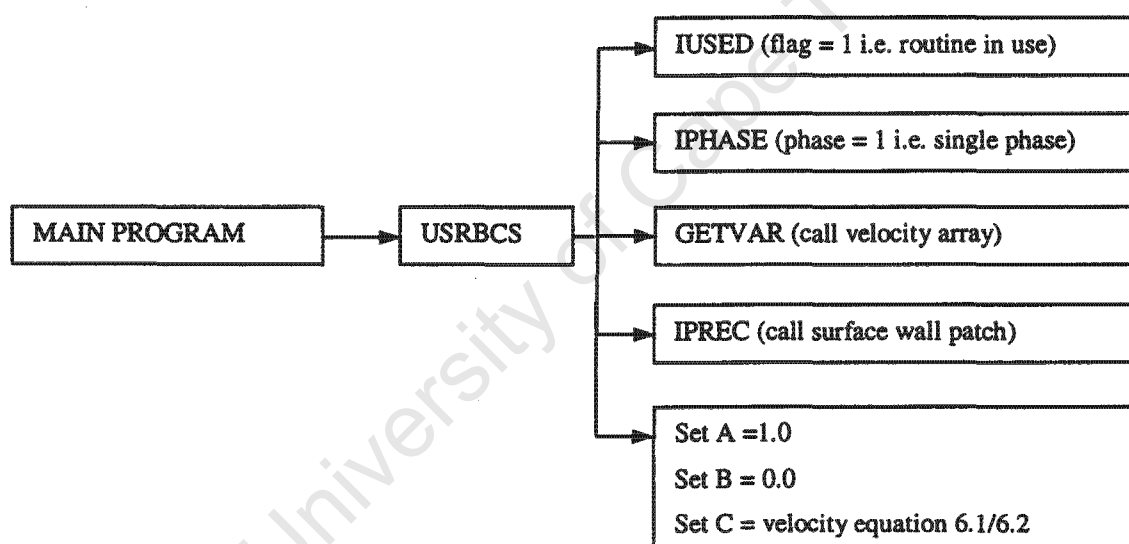


Figure 6.9 : Moving wall surface patch routine

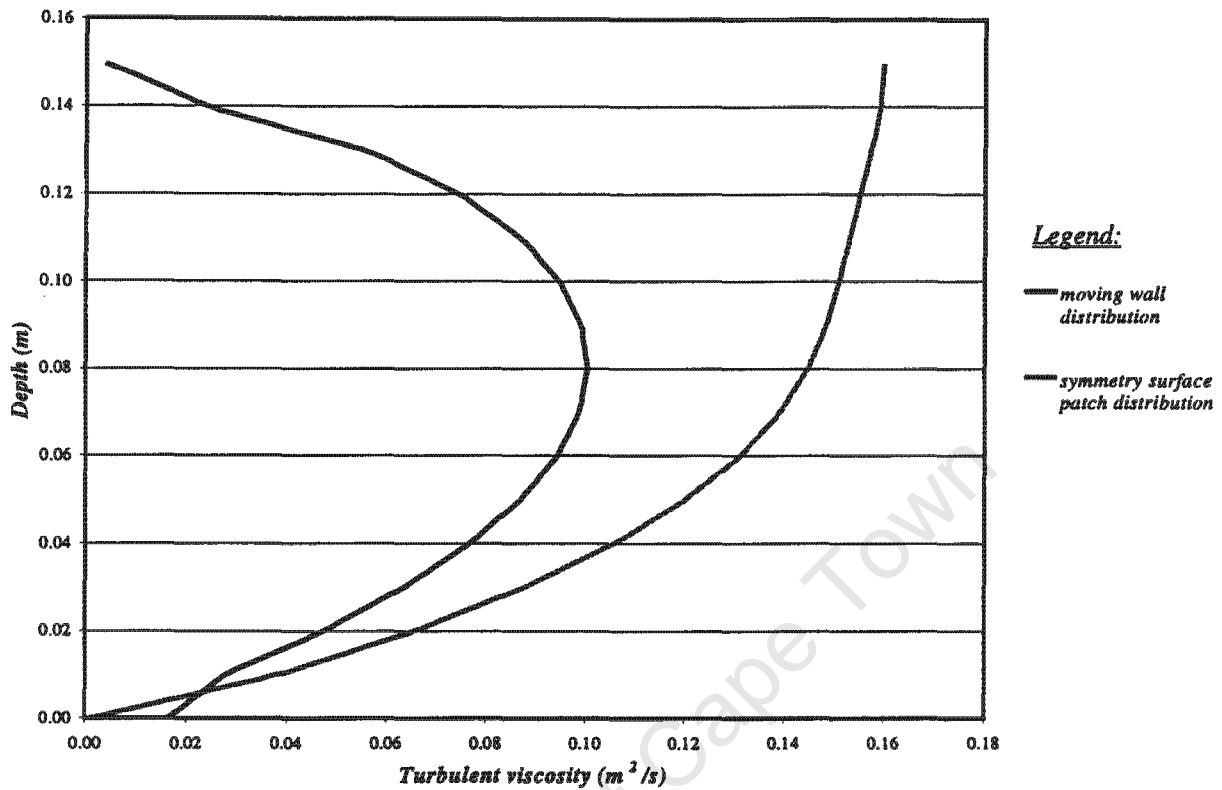


Figure 6.10 : Turbulent viscosity distribution with depth for moving wall and symmetry patch condition

6.4 Shear Stress Distribution with Depth

The shear stress distribution varies linearly with depth in laminar, transitional and turbulent flow. This was shown in Section 2.2. As power is a function of the shear stress, the program was calibrated against calculated shear stress distributions (equation 2.2),

$$\tau = \rho g S_o (D - y)$$

2.2

The shear stress is dependent on the turbulent kinetic energy k and rate of viscous dissipation ϵ , which are calculated at the inlet based on the given turbulent intensity

and dissipation length scale. This, however, results in a uniform distribution of these quantities at the inlet whereas in fully developed flow the distribution of k increases linearly towards the channel bed and the ε distribution is small at the surface increasing at an increasing rate towards the channel bed. As the shear stress is dependent on these quantities, the correct distributions had to be programmed in at the inlet patch.

Each model was initially run over a length of approximately 60 times the channel width, with a course mesh. The k and ε distribution at the outlet were then plotted in Microsoft Excel and a best-fit polynomial was used to describe these profiles. These distributions were then programmed in at the inlet patch of the actual model with the refined mesh (fig 6.11). In the three-dimensional case the distribution of these quantities at mid-height was plotted along with the vertical distribution to generate an equation describing the surface distribution at the inlet. These parameters were programmed in USRBCS along with the velocity distribution (Appendix, A6). This resulted in a linear shear stress distribution at the inlet patch.

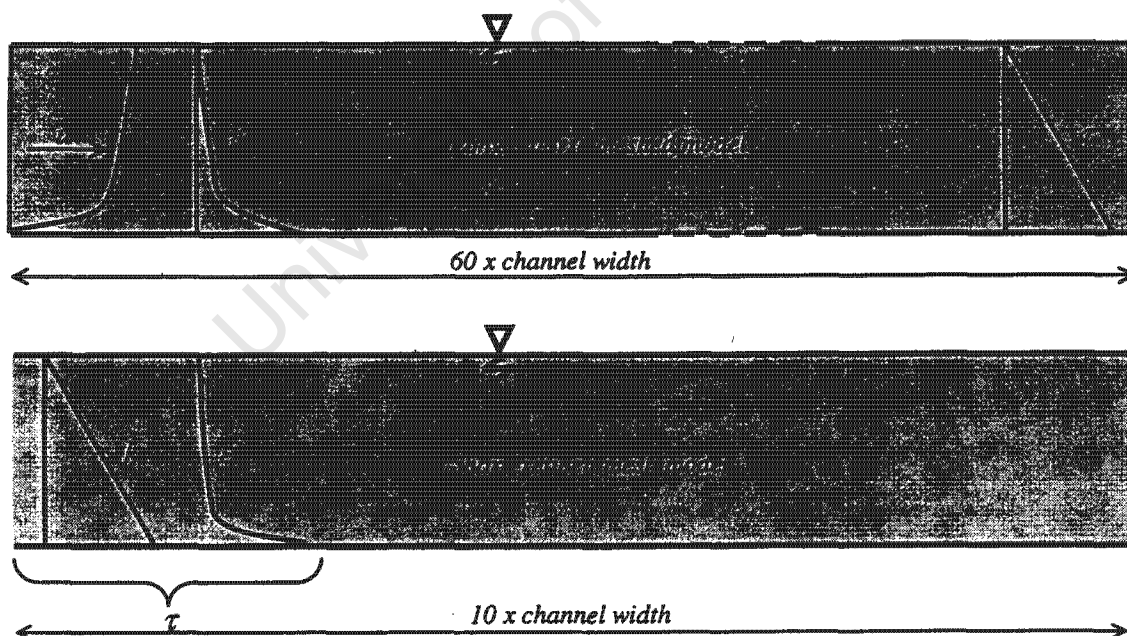


Figure 6.11 : Revised shear stress profile – setting k and ε

Once the shear stress was linear, the next aspect was the magnitude of this shear stress. For unidirectional, steady flow, in an infinitely wide, uniform channel the shear stress distribution can be calculated by hand (equation 2.2). The correct shear stress magnitudes were obtained for this simple case regardless of the selection of turbulence model. The more complex case of shear stress profiles over bedforms is discussed in Section 6.6.

6.5 Differencing Scheme Selection

The available differencing schemes are provided in Section 5.6. The selection of the differencing scheme was based on the modelling of a benchmark problem with widely accepted results. The problem selected was that of the lid-driven cavity. This is a two-dimensional problem, where the flow in a cavity is driven by a constant horizontal velocity over the surface, causing the flow to slowly rotate about some horizontal axis. Amongst the most well known benchmarking results are those produced by Chia et al. (1982).

Chia et al. (1982) used a multi-grid finite difference method for the streamline-vorticity formulations with an extremely fine, but uniform mesh. The standard model is a $1.0 \text{ m} \times 1.0 \text{ m} \times 0.1 \text{ m}$ cavity with a velocity across the lid of 1 m/s (Fig 6.12).



Figure 6.12 : Lid-driven cavity

This problem was run with the most commonly used differencing schemes:

- (i) CENTRAL
- (ii) HYBRID
- (iii) UPWIND
- (iv) QUICK

The vector plot from the QUICK scheme is shown in figure 6.13.

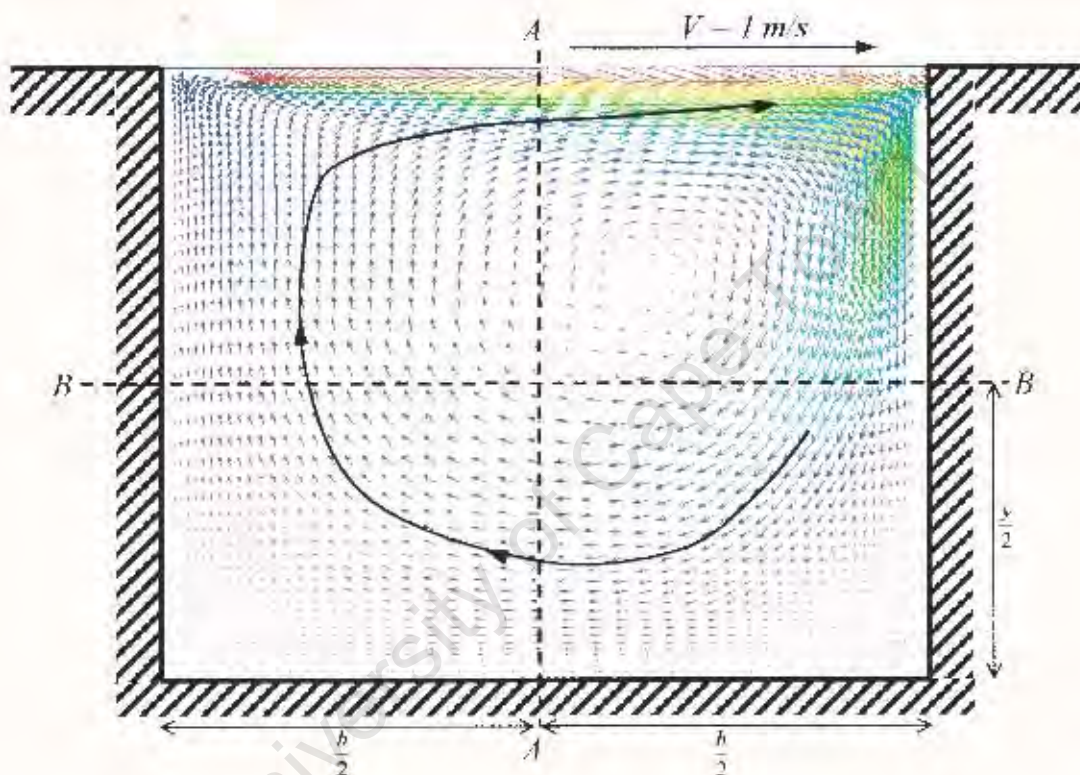


Figure 6.13 : Velocity vectors in a lid-driven cavity using the QUICK scheme

Figure 6.14 shows the variation for the horizontal velocity component across section A-A taken at mid-width across the cavity (fig 6.13) for the various schemes. These are compared to the data from Ghia et al. (1982).

The CENTRAL differencing scheme produced identical results to the HYBRID scheme. This implies that the Pe number (defined in Section 5.5) was most likely less than two i.e. the CENTRAL differencing scheme was used within the HYBRID scheme.

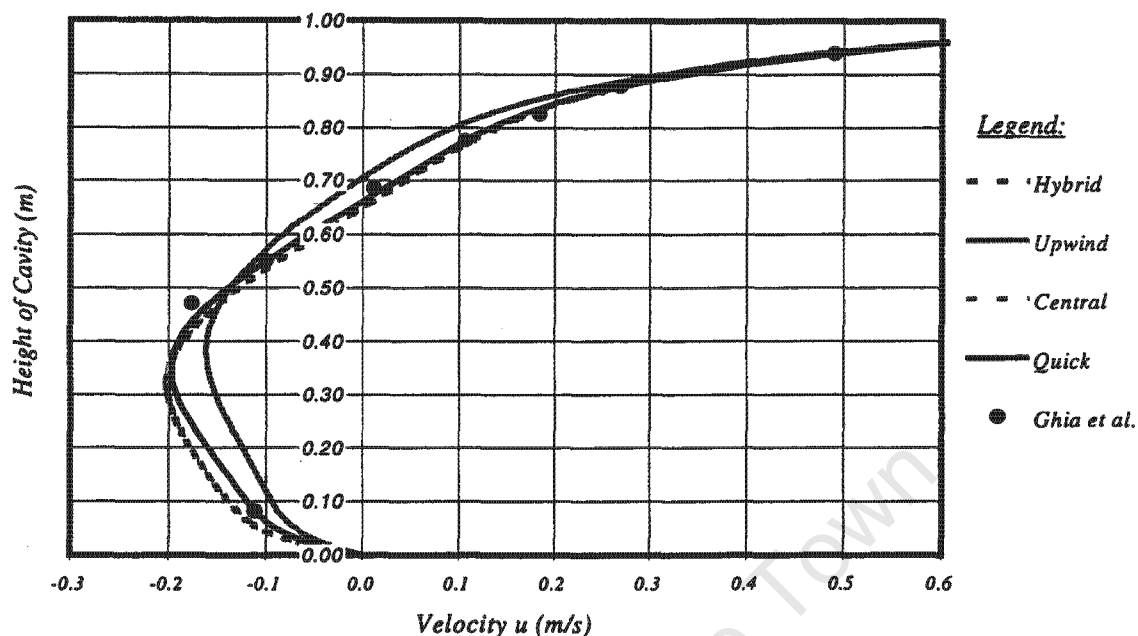


Figure 6.14 : The u -velocity variation with depth for a lid-driven cavity

Although the UPWIND scheme reduced the number of iterations and therefore the running time, it had the lowest velocities and the distribution differed from the other schemes (see fig 6.14). This was attributed to a bad choice of scheme since the flow direction in the cavity is not simply from left to right, but is from right to left in the lower regions and vertically oriented on the sides (see fig 6.13).

The QUICK scheme took the longest time to reach convergence, as expected since the order of accuracy is higher due to second order terms. The results, however, approximated those of Ghia et al. (1982) the most accurately.

Overall the QUICK scheme can be considered to be the most accurate for the lid-driven cavity case. The QUICK scheme was implemented throughout this open channel study for the velocity and pressure terms. However, the turbulent kinetic energy and rate of dissipation terms were modeled using either the UPWIND or HYBRID schemes. The reason for this is that the QUICK differencing scheme can give rise to converged solutions with negative k or ϵ values when the grid does not

adequately resolve the shear layer. The CFX Manual (1999) recommends the use of the UPWIND or HYBRID scheme for these terms and a higher order scheme e.g. QUICK for the velocity components to ensure positive values of k and ε .

6.6 Turbulence Criteria Selection

The turbulence input criteria include the dissipation length scale, the turbulence intensity and the choice of turbulence model. If one of the Reynolds stress models is selected, the number of initial eddy iterations should be set. There are further options to alter the empirically derived constants in the transport equations (Section 5.5.4) of specific models.

6.6.1 Length Scale, Intensity and Eddy Iterations

The dissipation length scale L and the turbulence intensity e are required to determine the inlet values of k and ε , i.e.

$$k = 1.5(eu)^2$$

6.4

$$\varepsilon = \frac{k^{1.5}}{0.3L}$$

6.5

where u is the average inlet velocity. The default value of e is given as $e = 0.037$. L is geometry dependent and a suitable choice is the hydraulic radius of the inlet, which is equivalent to the hydraulic depth for a wide channel.

The lid-driven cavity was used once again for a sensitivity analysis. The vertical velocity components were plotted across section B-B at mid-depth of the lid-driven cavity (see fig 6.13) for a variety of values of the intensity and dissipation length scale (fig 6.15 & 6.16 respectively).

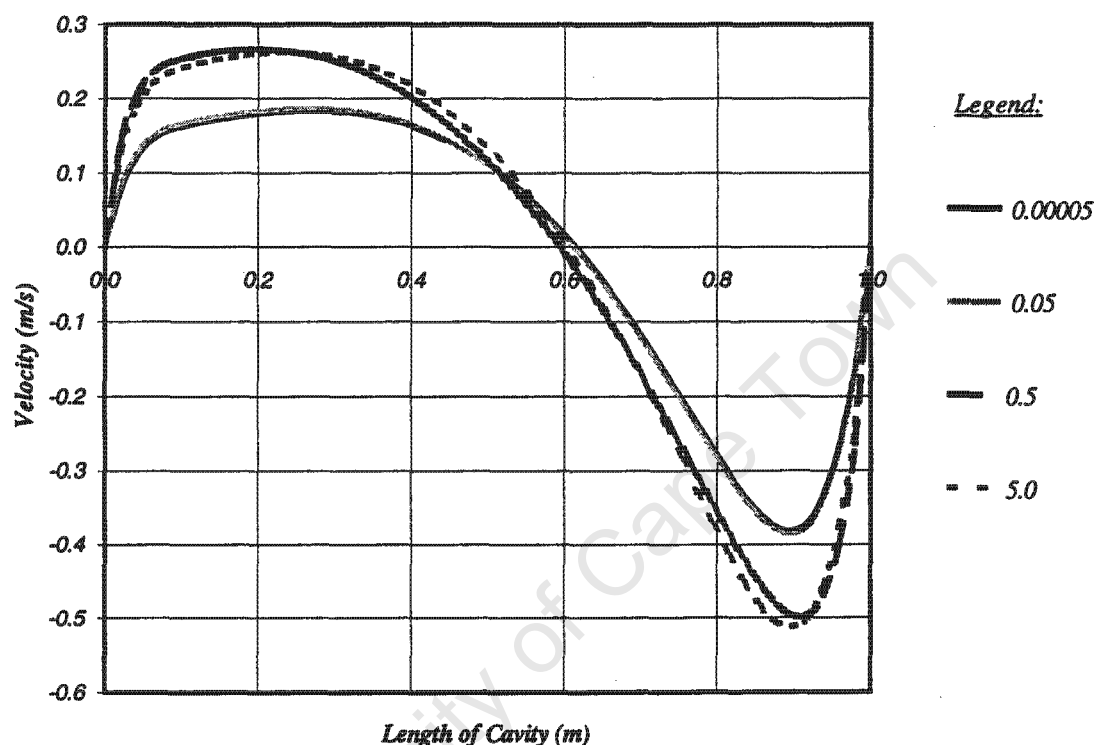


Figure 6.15 : v -velocity distribution for various turbulent intensities

It can be observed that at low intensities such as 0.05 and 0.00005 the predicted velocity distributions are similar. The maximum and minimum velocity values fall well within the range of Ghia et al. (1982) i.e. -0.52 to 0.37 m/s, however anecdotal data suggest that further mesh refinement to reduce the cell sizes results in the velocity distribution approaching that of Ghia et al. (1982) more closely. The velocity results were relatively insensitive to changes in L from 0.01 to 10.

It has been found advantageous, for convergence purposes, to start the Reynolds stress or flux models from an approximate solution provided by an anisotropic eddy viscosity model (CFX manual 1999). The initial eddy viscosity iterations are

therefore set to specify the number of iterations of the eddy viscosity model to be performed before the Reynolds stress or flux model is started. The default value is 100. A sensitivity analysis showed no change in solution for values up to 150.

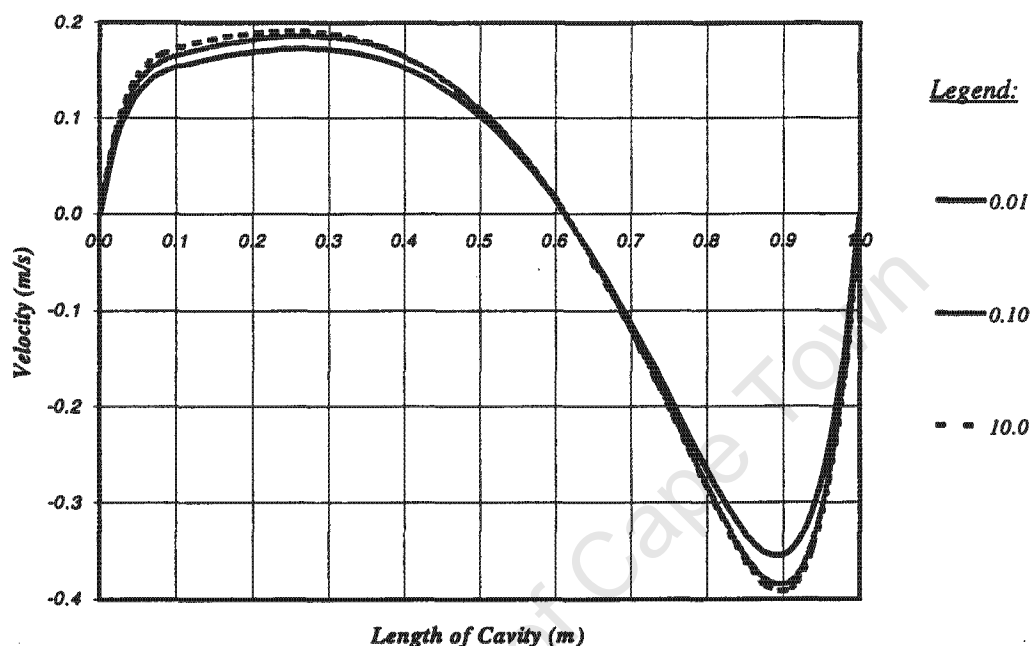


Figure 6.16 : v -velocity distribution for various dissipation length scales

6.6.2. Selection of Turbulence Model

For uniform, steady flow conditions in a rectangular channel, the selection of turbulence model did not impact the shear stress distribution. An analysis was carried out on the shear stress distribution with depth over bedforms using the published data of Cellino and Graf (2000). Cellino and Graf (2000) provided experimental data of the variation of the velocity and the Reynolds stresses with depth for a two-dimensional, sediment-laden flow over bedforms fixed at 2 and 3 degrees respectively (fig 6.17).

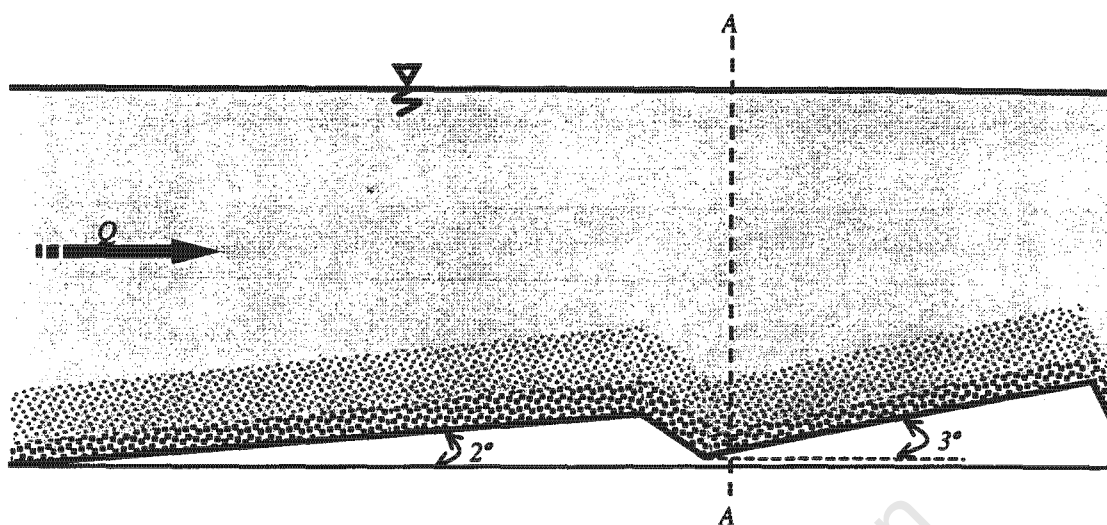


Figure 6.17 : Two-dimensional flow over bedforms

Figure 6.18 shows how the mesh was refined over the bedform and in the region of A-A for this purpose.

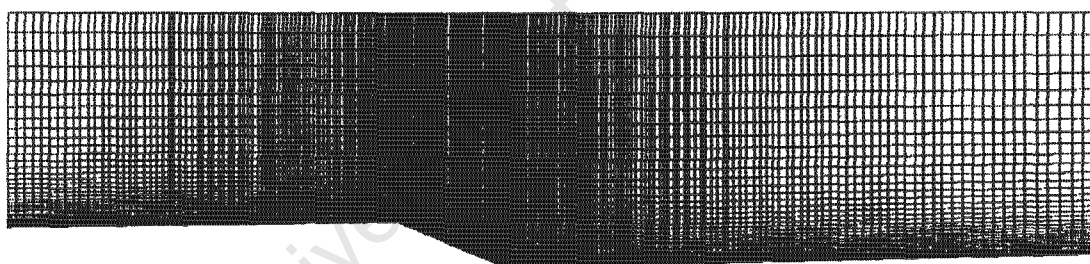


Figure 6.18 : Mesh refinement around the change in geometry and section A-A

The velocities and stresses for the various models were plotted along section A-A. Of the turbulence models, only the algebraic RSM and the $k-\varepsilon$ model provided predictions close to that of the experimental data. The horizontal velocity component divided by the average velocity and the dimensionless shear stress are plotted against the dimensionalised depth in figure 6.19 and 6.20 respectively for these two models.

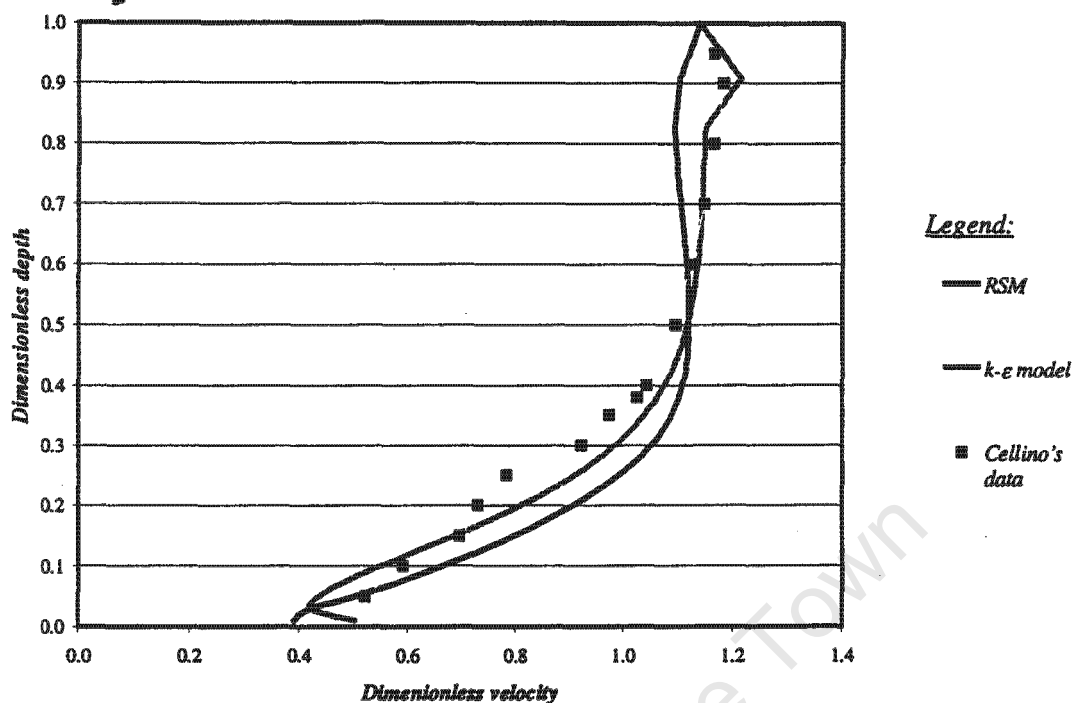


Figure 6.19 : Variation of dimensionless velocity $\frac{u}{u}$ with dimensionless depth $\frac{y}{D}$

The $k-\epsilon$ model agrees more closely with the experimental data in this instance, although none of the models predict the stress particularly well. Based on this result, the two-dimensional problems were modelled using the $k-\epsilon$ model. The $k-\epsilon$ model has been known to over predict the shear stress, whilst the low Reynolds Number $k-\epsilon$ model was found to predict the stresses most accurately. The low Reynolds Number $k-\epsilon$ model was thus implemented on moving to the three-dimensional analysis with vortex shedding.

Experimental results were available for each run throughout this study. The physical model studies carried out (e.g. Babaeyan-Koopaei 1996; Harnett 1998; Midgley 2000; Mitchell 2000) typically involved Reynolds Numbers in the range 10 000 – 20 000, for which the low Reynolds Number $k-\epsilon$ model is advised. Implementing this model required an extremely fine mesh at the channel bed as no wall functions are incorporated into this turbulence model.

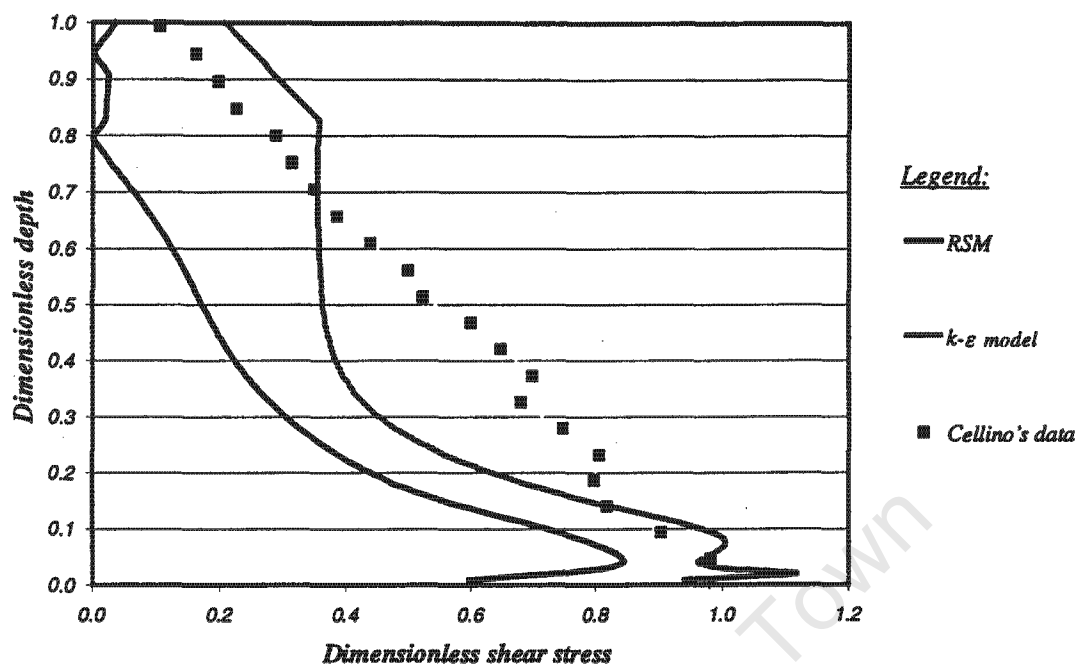


Figure 6.20 : Variation in dimensionless shear stress $\frac{\overline{u'v'}}{u_*^2}$ with dimensionless depth $\frac{y}{D}$

6.7 Pressure Correction

The available pressure correction algorithms in CFX-V4.3 (1999) are:

- (i) SIMPLE
- (ii) SIMPLEC
- (iii) PISO
- (iv) Non-iterative PISO

The basis for updating pressure and correcting velocity components is the SIMPLE algorithm. The SIMPLEC is a variation of the SIMPLE that differs in its derivation of the momentum equation. The SIMPLEC requires a small amount of extra computing work, so the CPU cost is negligible. It has proved less sensitive to the

selection of the under-relaxation factors and in general requires less under-relaxation thereby speeding up the solution time.

In the PISO algorithm, a second pressure-correction equation is solved in order to improve the solution of the momentum equations. The two pressure-correction steps increase the computational cost per iteration of the PISO algorithm as compared with the SIMPLE algorithm. The PISO algorithm is however less approximate and usually results in a reduction in the number of outer iterations for convergence (CFX Manual 1999).

The non-iterative PISO is a semi-implicit non-iterative transient algorithm where only one iteration is performed at each time step. This algorithm differs from the PISO in that it includes some boundary condition information in the matrix structure, so that the boundaries are implemented implicitly rather than in the usual explicit manner.

The default pressure correction is the SIMPLER algorithm. This was used throughout, and only altered when convergence difficulties were encountered. The SIMPLE and PISO algorithms were used in these simulations.

6.8 Solvers

The available solvers were outlined in Section 5.7. In general the default solvers were used (table 6.1). These solvers were adequate for the two-dimensional problems, however the solutions diverged for the three-dimensional abutments and piers. In these cases, the BLOCK STONE solver was implemented for the u , v , and w variables and in the case of the abutment this proved successful.

In the case of the piers, when the BLOCK STONE solvers were used with different pressure corrections (Section 6.7) the residuals diverged. The CFX Manual (1999) recommended the AMG solver for the pressure term in cases when it is thought that the pressure-correction was not being adequately treated. Furthermore, if the percentage of well-ordered blocks is low, which is the case with the pier, the AMG or

BLOCK STONE solver should be used to solve for the pressure term (CFX Manual 1999). The AMG solver was therefore introduced for the pressure term and convergence was attained.

Variable	Solver
u -velocity	STONE
v -velocity	STONE
w -velocity	STONE
pressure	ICCG
turbulent kinetic energy	LINE SOLVER
rate of viscous dissipation	LINE SOLVER

Table 6.1 : Default variable solvers

6.9 Buoyancy

The geometry creation can be facilitated by the creation of a horizontal geometry and hence specifying a gravitational acceleration vector that accounts for the bed slope. This requires the use of the buoyancy option and the heat transfer switch had to be activated. Although the temperature variation in open channel flow is negligible, the buoyancy parameter requires a reference temperature. The fluid, water in this instance, had to be defined as a standard fluid with a temperature of 293 K. This buoyancy parameter was implemented for all applications where the bed slope was not horizontal e.g. the vertical weir simulation (Harnett 1998).

6.10 Steady and Transient Flow

The transient option was initially investigated and later rejected for flows where the solution generally varies as a function of time. These cases included the abutments and the piers as vortices are shed from separation points. The time-stepping options in CFX-V4.3 (1999) include fixed or adaptive time stepping, or a combination of the

two. Fixed time stepping involves defining a fixed time interval for each step and adaptive time stepping provides an automatic control of the time step so that convergence is achieved with the maximum efficiency.

The fixed time-stepping option requires user input regarding the number of time steps, the length of the time-step, the initial time, the selection of either implicit (backward) or Crank Nicolson (central) time differencing schemes, and the selection of either linear (first-order) or quadratic (second-order) time differencing.

The CFX Manual (1999) advises against the simultaneous use of the Crank Nicolson scheme and the quadratic time differencing. The default backward differencing and linear time differencing were thus selected and no problems were encountered.

The adaptive time-stepping option requires the following input:

- (i) the number of time steps
- (ii) the initial time step
- (iii) the minimum time step
- (iv) the maximum time step
- (v) a multiplication factor
- (vi) a division factor
- (vii) the minimum interval between increments
- (viii) the maximum number of contiguous decrements
- (ix) implicit or Crank Nicolson time differencing scheme
- (x) linear or quadratic time differencing
- (xi) initial time
- (xii) maximum time

These parameters were set at default values where possible. The minimum time step and the maximum time step played an important role in guiding the solution and the rate of convergence. The minimum time step was selected to be sufficiently small to identify changes in a short space of time, but not so small as to cause a major increase

in computational time. Typically values of 0.0001 seconds were used. The maximum time step was set as 5 seconds.

The modelling of the vortex shedding around a rectangular cylinder revealed that implementing the fixed time stepping initially and then restarting the file with adaptive time stepping reduced the CPU time and gave more accurate results of the drag coefficient in the study of flow around a rectangular cylinder.

The abutment was modelled using this approach. The solver was left for 24 hours and then physically stopped, as convergence had not yet been attained. The predicted velocities showed no sign of a vortex street. This was attributed to the damping of the eddy path by the moving wall assumption at the free surface. It was found that when the solver was stopped at successive intervals, the predicted power distribution seemed to be approaching the steady state solution. Since there was no apparent difference between the channel bed power distribution generated by both alternatives, the steady state option was implemented.

6.11 Topography Adaptation

The adaptation of the Movability Number to incorporate the angle of repose for sediment particles situated on a slope, was discussed in Section 3.5. The resulting equation was

$$\left. \frac{\mathbf{u}_*}{V_{ss}} \right|_{\beta, \gamma} = \psi \left. \frac{\mathbf{u}_*}{V_{ss}} \right|_0$$

3.48

where the suffix '0' implies a horizontal bed, i.e. the Movability Number equal to 0.17 for incipient motion. β and γ are the variation of the velocity ratio with the longitudinal and transverse bed slope respectively and $\mathbf{u}_*$ is the shear velocity vector with the direction of the bed shear stress. ψ is given by

$$\psi = \sqrt{\cos\beta \left(1 - \frac{\tan\beta}{\tan\phi_s}\right) \cos\gamma \left(1 - \frac{\tan^2\gamma}{\tan^2\phi_s}\right)^{0.5}}$$

3.49

where ϕ_s is the angle of repose for the particular sediment type.

Equation 3.48 was used to predict the equilibrium conditions for flow around a pier. The experimentally observed scour hole profile around a pier (Babaeyan-Koopaei 1996) was created in CFX. Equation (3.48) for the critical motion condition could then be subtracted from the actual condition (Section 6.2) given by the Movability Number adjacent to the channel bed such that the scour potential Ω is given by:

$$\Omega = \frac{u_*}{V_{*c}} \Big|_{\text{Actual}} - \psi \frac{u_*}{V_{*c}} \Big|_0$$

6.6

A negative value of Ω implies that deposition is expected, a zero value for Ω would imply equilibrium conditions i.e. no scour or deposition and a positive value of Ω implies scour conditions.

Any variables required other than those given by CFX should be programmed in user routine USRTRN (Section 6.2). The difficulty encountered in programming equation 3.49 was the determination of the longitudinal slope β and the transverse slope γ .

The longitudinal slope β was found by determining the angle that the velocity vector at the boundary makes with a horizontal plane. The assumption is that the velocity vector is approximately parallel to the boundary in the cell adjacent to the boundary. β is measured as positive down from the horizontal plane, which implies that a particle situated on a downstream slope will be more likely to move than if it were on a horizontal bed, while a particle on an upstream slope will have more resistance to movement. β is given by:

$$\beta = \arctan\left(\frac{-v}{\sqrt{u^2 + w^2}}\right)$$

6.7

Determining this equation involved specifying a user three-dimensional patch including all the boundary cells adjacent to the channel bed. Utility routine IPALL is then used in USRTRN to select the user three-dimensional patch cell centres to determine β . In the rare case where the u - and w -velocity components are both equal and in the range $-10^{-12} < u/w < 10^{-12}$, i.e. effectively zero, the denominator is set as 10^{-12} as β is ± 90 degrees, e.g. downflow adjacent to a pier, or zero (if the v -velocity component is also zero).

The transverse slope γ is the angle between the transverse vector 't' in the plane of the boundary and the horizontal plane. This t-vector is perpendicular to the velocity vector i.e. transverse to the direction of the flow and is calculated from the cross-product of the normal vector to the boundary plane $\mathbf{n}(n_x, n_y, n_z)$ and the velocity vector $\mathbf{U}(u, v, w)$, i.e. t is given by

$$\mathbf{t}(t_x, t_y, t_z) = (vn_z - wn_y, wn_x - un_z, un_y - vn_x)$$

6.8

The transverse slope γ is then calculated from

$$\gamma = \arctan \frac{t_y}{\sqrt{t_x^2 + t_z^2}}$$

6.9

The normal vector to a plane is not readily available in CFX. However, each cell stores three area vectors that give the direction of the cross-sectional area of the cell at the cell centre. It is possible to interrogate this array and extract the area vector of the face parallel to the relevant boundary, i.e. the channel bed. This area vector points away from the boundary and its magnitude is related to the area of the cell. There is

no need for this vector to be normalised, as only the direction is required. To extract this vector, a user two-dimensional patch was defined while creating the geometry. The boundary nodes at the centre of each boundary cell face were called in USRTRN using utility routine IPALL.

The node numbering system is such that the boundary nodes are numbered similarly to the cell centres. It is thus possible to loop through the user two-dimensional patch and extract the area vector, which can be stored in a locally defined array. β , t , γ are then computed from the velocity vectors obtained from the 3D patch adjacent to the boundary 2D patch. Finally the scour potential Ω is computed (equation 6.6) and the values stored at the 3D patch cell centres.

The post-processors in CFX interpolate the values between cell centres. As the values are only calculated in the user patch adjacent to the channel bed, the results cannot be presented graphically using the built-in post-processors. The results were therefore written to an alternative file in a format that could be interpreted by IDL (2000). This package provided a two-dimensional spatial representation of the values for the scour potential Ω (equation 6.6) over the equilibrium scour profile.

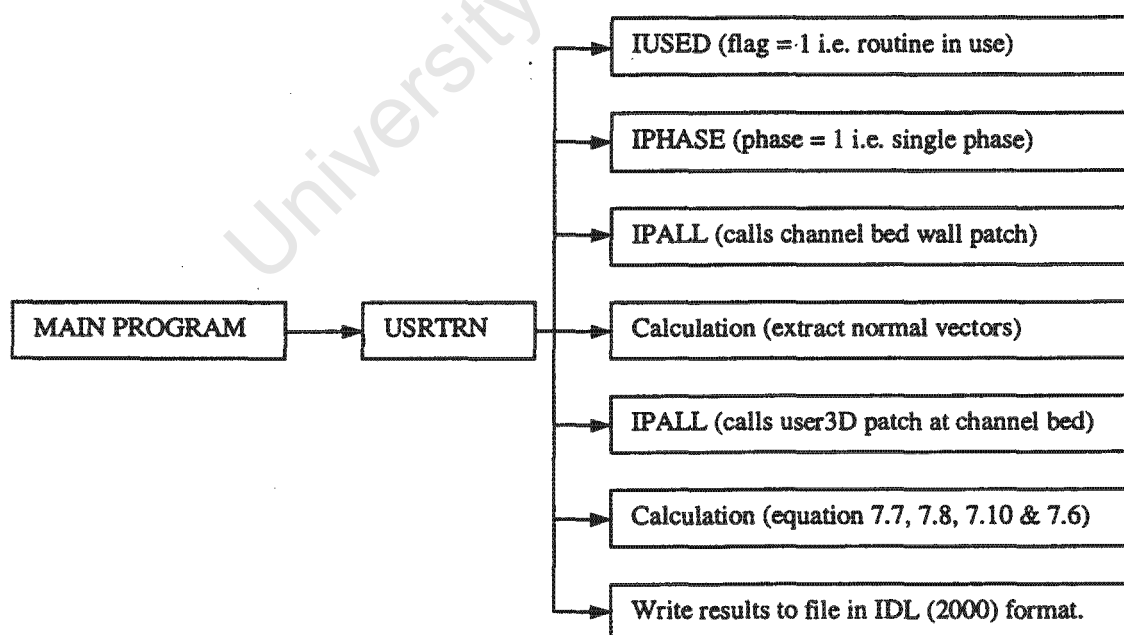


Figure 6.21 : Subroutine for determining the topography adaptation incipient motion function

This subroutine is described in figure 6.21 (see routine in Appendix, A7). This program was only implemented for the analysis of the scour hole profile around a cylindrical pier that was measured experimentally by Babaeyan-Koopaei (1996).

6.12 Final Methodology

Once the software CFX-V4.3 (1999) had been adequately calibrated for the modelling of open channel flow, it was possible to put together a methodology for the various flow problems. The procedure is outlined by means of a flowchart (fig 6.22).

This methodology was implemented for a variety of problems. The applications include the transport of beads over a weir and under an underflow structure (Offerman 1998), as well as through a sudden contraction (Feuilloley 1999) and a sudden expansion (Lyons 1999). The scour prediction includes sediment profiles upstream of a weir (Harnett 1998), around an abutment (Mitchell 2000) and in the vicinity of a pier (Babaeyan-Koopaei 1996; Midgley 2000). Chapter 7 and 8 describe the results of the bead and sediment applications respectively.

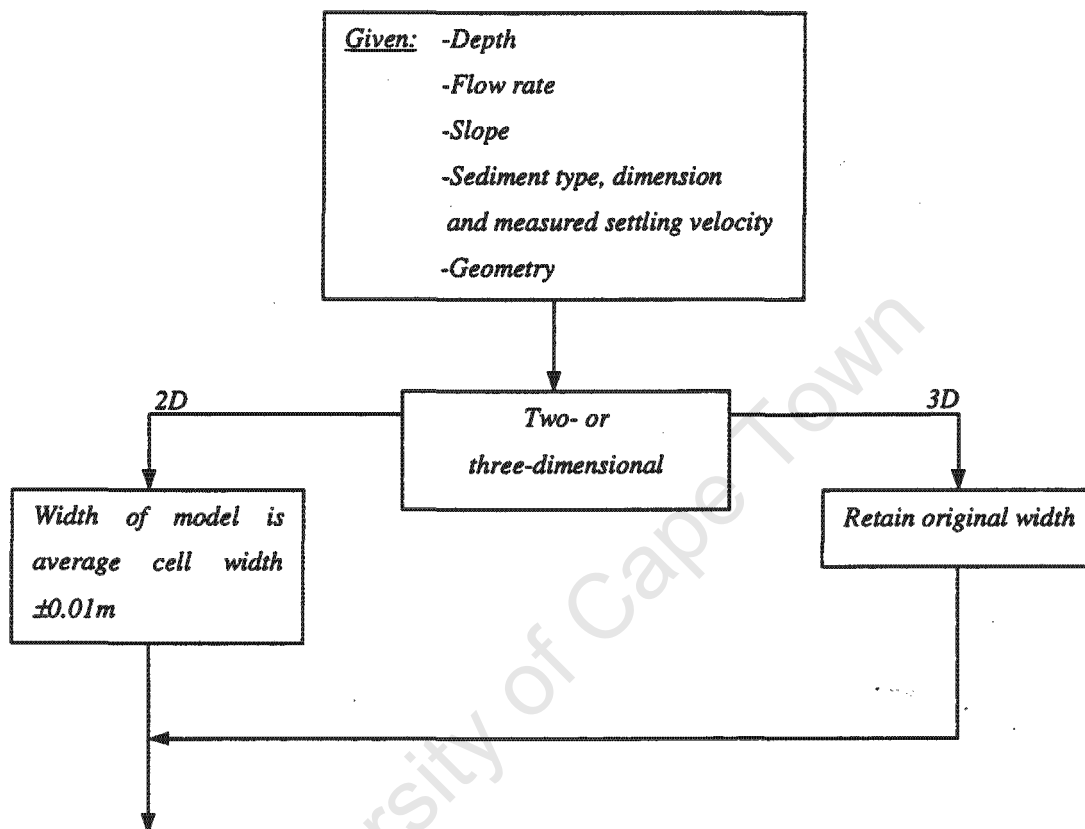


Figure 6.22 : Final methodology flowchart

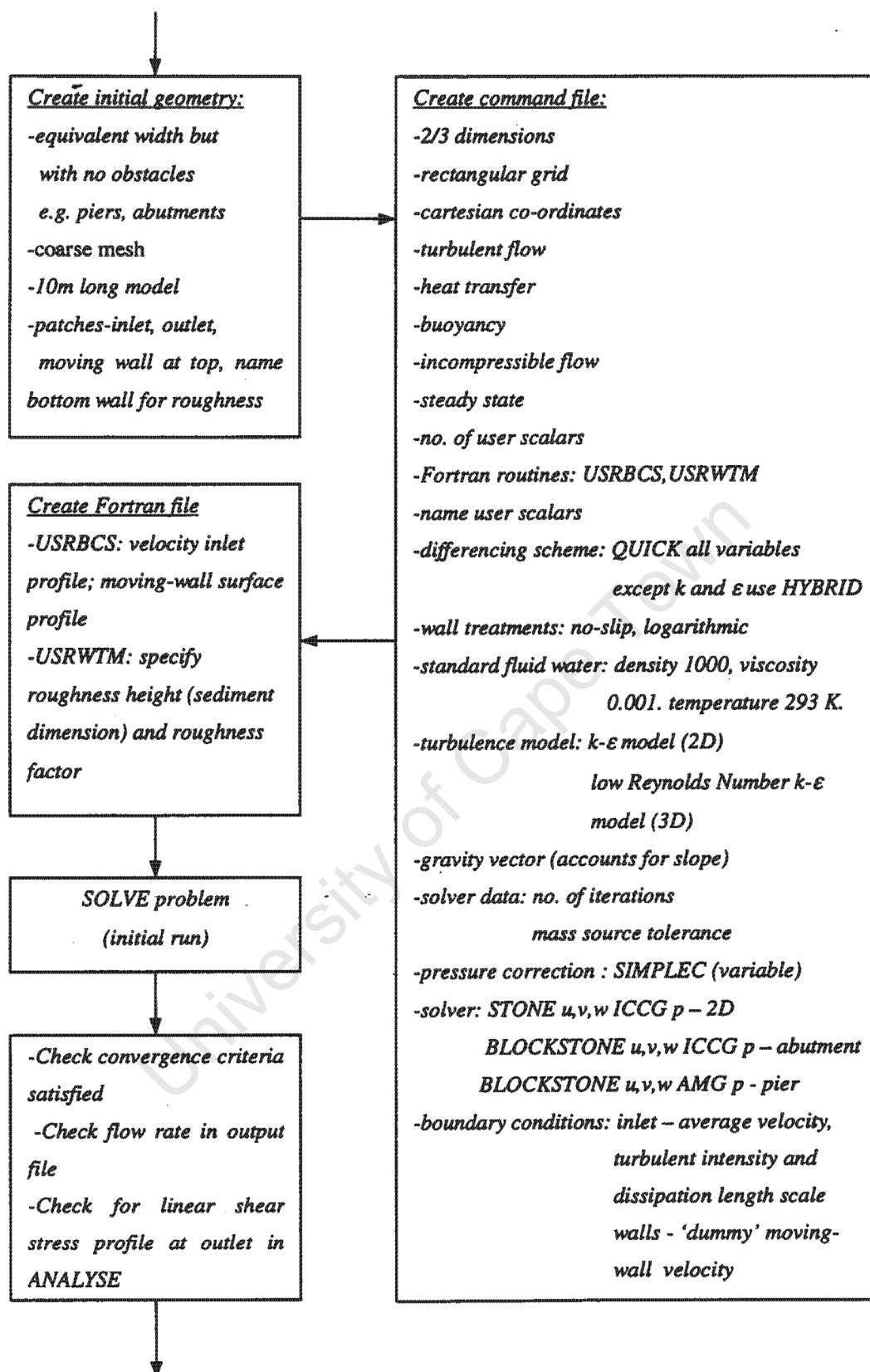


Figure 6.22 : Final methodology flowchart

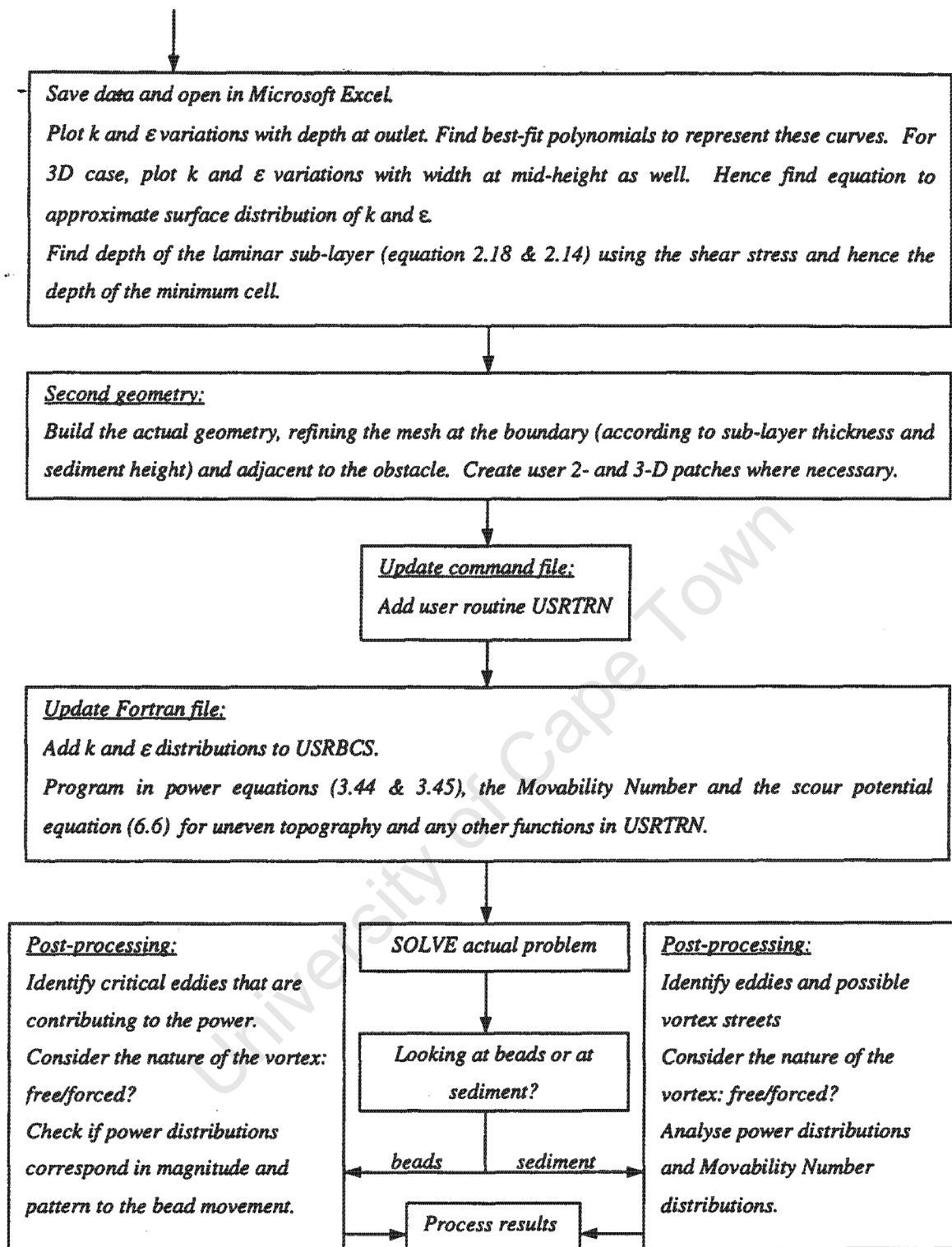


Figure 6.22 : Final methodology flowchart

Chapter 7

Numerical Predictions of the Bead Movement

This chapter gives a detailed description of the numerical predictions of bead movement over a vertical weir, under an underflow structure and through a sudden contraction and expansion. A comparison is made with the actual observed movement. The numerical model is a tool that facilitates the computation of the power distribution within the fluid domain. It was also used to determine the flow patterns around the structures and the critical eddies contributing to the applied stream power.

7.1 Movement over a Vertical Weir

Offerman (1998) investigated the movement of beads over a vertical weir in a rectangular channel for various flow rates and weir heights. The experiments were carried out in a horizontal flume that was 6 m long, 310 mm wide and 300 mm deep and consisted of glass walls and a copper floor. Flow guides were placed upstream of the weir and plastic vanes were positioned at the obstacle to reduce mixing in the horizontal plane (fig 7.1).

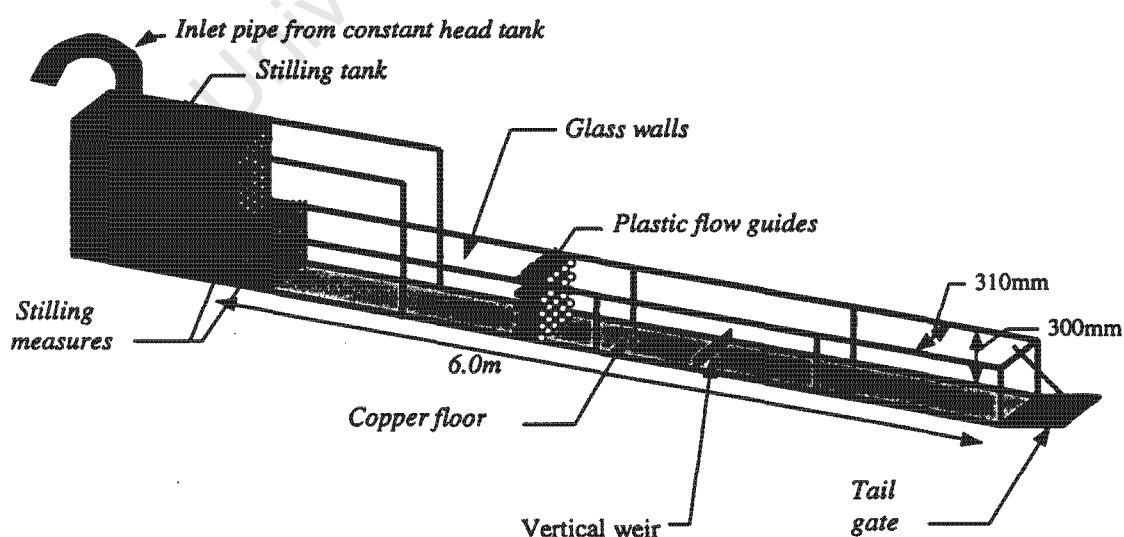


Figure 7.1 : Offerman's (1998) laboratory flume

Offerman (1998) released beads in the upstream section and observed whether they were trapped by the weir or passed over it. The bead densities, settling velocities and individual power requirements are summarised in table 7.1.

Bead description:	V_s (mm/s)	ρ_r (kg/m ³)	P_r (W/m ³)
Brown rubber	31	1020	6
Yellow plastic	80	1050	35
Red rubber	110	1240	256
Black rubber	172	1440	733

Table 7.1 : Description of Offerman's (1998) beads

Three of these experiments (Offerman 1998) were simulated using the numerical model. The flow details for the experiments and the corresponding numerical models are summarised in table 7.2.

Model no.	Exp. no.	Exp. flow rate (l/s)	Exp. Width (mm)	Model flow rate (l/s)	Model width (mm)	Weir height (mm)	Water depth (mm)	Particles passing over weir			
								brown	yellow	red	black
								%	%	%	%
A1	4 (test 1)	7.63	310	0.246	10	53	150	100	0	0	0
A2	6 (test 1)	2.20	310	0.071	10	31	63	100	50	0	0
A3	10(test 1)	2.83	310	0.091	10	69	97	100	35	0	0

Table 7.2 : Experiment and numerical model data for Offerman's (1998) experiments

The numerical model results were similar for these three runs, and thus only one example will be illustrated here (model no. A1, table 7.2). All the results are summarised in table 7.3.

Figure 7.2 illustrates the numerical model geometry for model A1. The sides were modeled with a symmetry condition i.e. a two-dimensional analysis. The width was therefore reduced to conserve cells, which reduced the model flow rate to 0.246 l/s.

The experimentally measured height of the water over the weir was 94.3 mm. As the change between the upstream depth and the water height over the weir was approximately 1.4 % of the total depth, it was considered negligible. The water surface was therefore modeled as a horizontal wall, parallel to the channel bed, moving at the surface velocity of the water. The outlet condition was specified as the region above the weir, as only the power distribution upstream of the weir was required.



Figure 7.2 : Typical dimensions of the flow domain prior to a vertical weir

Figure 7.3 is a vector plot of the velocity directions prior to the weir. The flow direction is from left to right. The velocity magnitudes correspond with those measured in the laboratory experiment, the maximum and minimum velocities depicted by the red and blue colours respectively. The logarithmic velocity distribution is well developed prior to the weir. Adjacent to the weir, the flow is slowed substantially in the lower region, while accelerating over the weir in the upper region. This steep velocity gradient results in a virtual eddy prior to the weir.

The unit stream power distribution was generated using the dissipation function (Section 3.4.6). Figure 7.4 shows this power distribution in the region of the weir. As the only particles that were trapped were the brown rubber beads, the maximum unit stream power region has been set at $\geq 6 \text{ W/m}^2$.

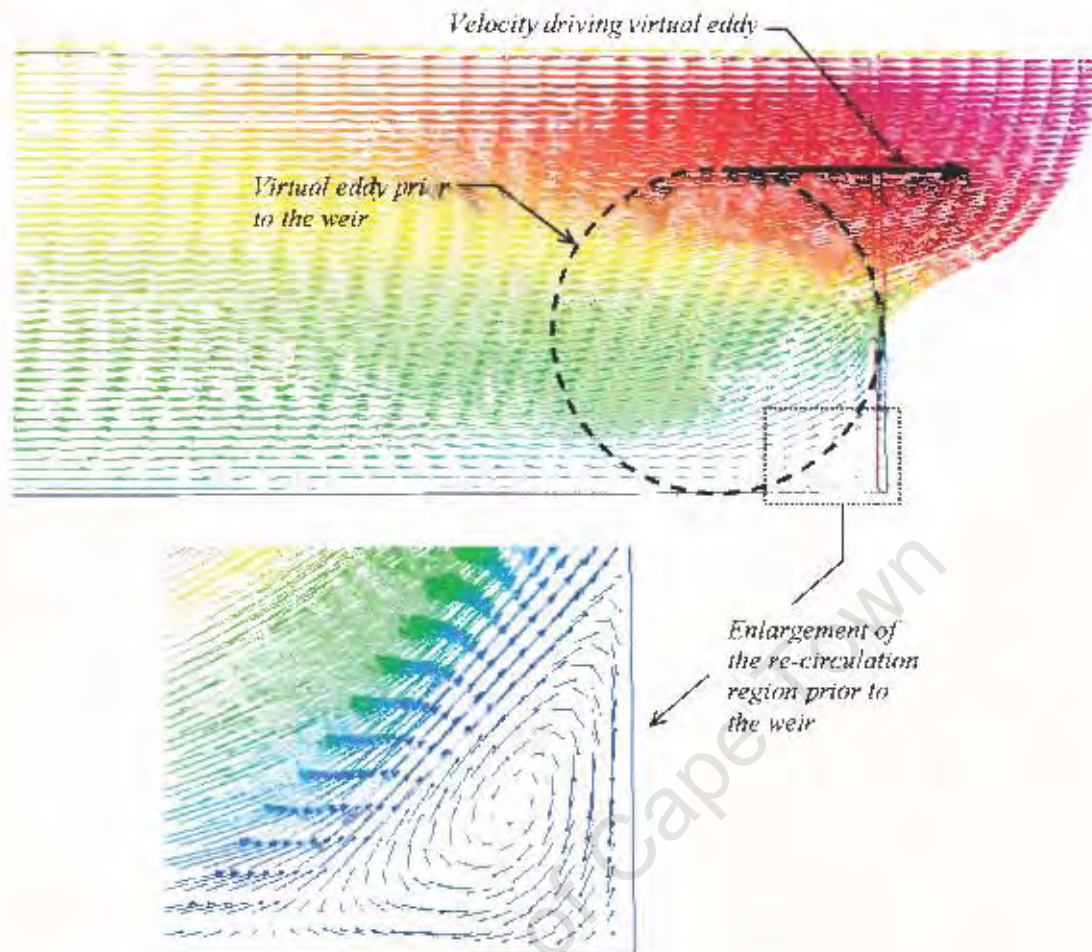


Figure 7.3 : Vector plot of velocity prior to the weir

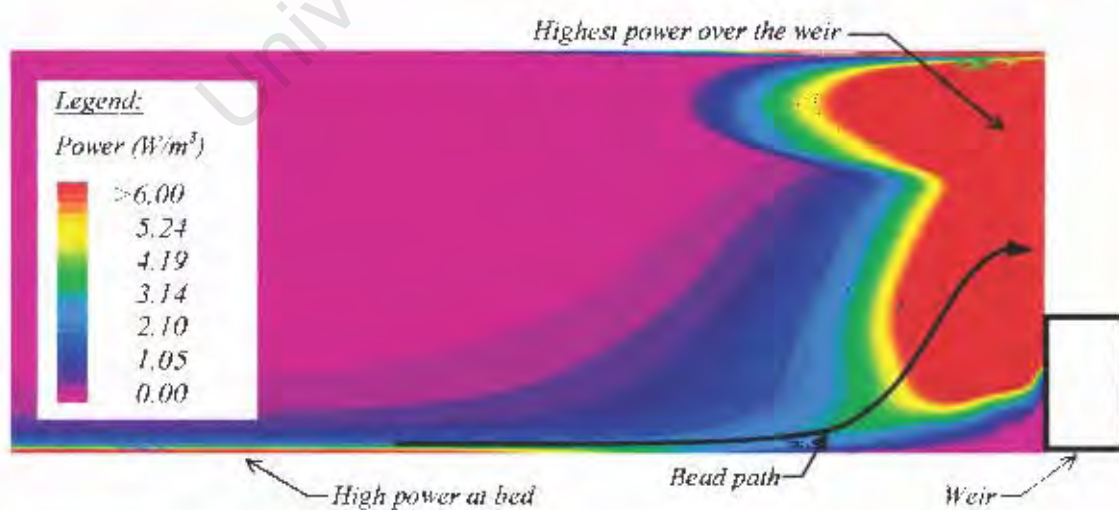


Figure 7.4 : Unit stream power distribution prior to the weir

The red region of high power adjacent to the bed should extend to the red section over the weir for the beads to pass over the weir. The reason the beads were not trapped could be attributed to the momentum of the individual beads. This momentum would cause the beads to cross the area of lower power between the two red regions, as once suspended the bead tends to move along a particular streamline. This distribution does explain the reason the remaining beads did not pass over the weir, since their power requirements considerably exceed those available from the applied unit stream power of the water. The high power distortion along the surface is due to the free surface being modeled as a moving wall. This raises the local velocity gradients thus greatly raising the local stream power over the actual stream power value.

Table 7.3 summarises the maximum power values over the weirs and the value of the minimum stream power zone linking the bed region to the power at the weir for the three vertical weir cases that were numerically simulated. The power requirement of the largest bead to pass over the weir in Offerman's (1998) physical experiments is also shown.

Model no.	Maximum power over weir (W/m ³)	Linking stream power zone (W/m ³)	Power requirement of largest bead to pass over the weir (W/m ³)
A1	53	2.1	6
A2	103	1.5	35
A3	443	0.8	35

Table 7.3 : Maximum power value and linking power zone at the weir

It can be observed that in general the linking stream power zones are much lower than that which is required to move the beads.

7.2 Movement under an Underflow Structure

Offerman (1998) investigated the movement of beads under an underflow structure or 'suspended sill' as he termed it. These experiments were conducted in the same flume as before (figure 7.1, Section 7.1). The principal objective was to observe

whether floatables passed under the underflow structure or whether they were trapped at the surface. Offerman (1998) used Compion's (1997) criteria to predict the bead movement. A critical eddy prior to the sill had to be identified in order to apply Rooseboom's (1992) rigid body analogy. It was found that this eddy extended from the water surface to the depth of the sill below the surface (fig 7.5). The power requirement to suspend a bead could be compared to the stream power dissipation resulting from the underflow structure.

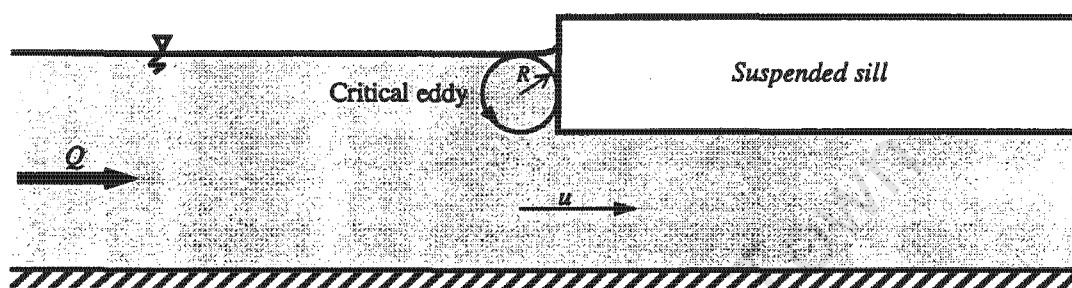


Figure 7.5 : Eddy prior to the suspended sill (Offerman 1998)

The selected beads were less dense than water and had negative settling velocities. The bead densities, settling velocities and individual power requirements are summarised in table 7.4.

Bead description:	V_{st} (mm/s)	ρ_b (kg/m ³)	P_r (W/m ³)
Cork	-214	484	1090
Polystyrene	-305	22	2960

Table 7.4 : Description of Offerman's (1998) floatable beads

Offerman (1998) investigated the bead movement for eleven different combinations of flow rate and underflow structure height. Ten of these experiments were simulated numerically. The flow parameters for the ten experiments and the corresponding numerical models are summarised in table 7.5.

Model no.	Exp. no.	Exp. flow rate (l/s)	Exp. width (mm)	Model flow rate (l/s)	Model width (mm)	Sill height (mm)	Water depth (mm)	Particles passing under sill	
								cork	polystyrene
								%	%
B1	25	4.3	310	0.140	10	44.9	50	70	0
B2	26	5.0	310	0.162	10	44.9	56	0	0
B3	27a	8.8	310	0.283	10	66.0	78	65	0
B4	27b	8.8	310	0.283	10	66.0	85	0	0
B5	28	11.1	310	0.036	10	66.0	87	20	0
B6	29a	13.7	310	0.441	10	88.1	98	100	60
B7	29b	13.7	310	0.441	10	88.1	104	70	0
B8	30a	10.1	310	0.327	10	88.1	96	100	0
B9	30b	10.1	310	0.327	10	88.1	100	0	0
B10	31	6.8	310	0.220	10	88.1	95	0	0

Table 7.5 : Experiment and numerical model data for Offerman's (1998) experiments

Ten of Offerman's (1998) experiments were simulated using CFX-V4.3. As the numerical model results were similar, one case will be illustrated (model no. B8, table 7.5) and the remaining nine cases will be represented in a table. The geometry for this model is shown in figure 7.6.

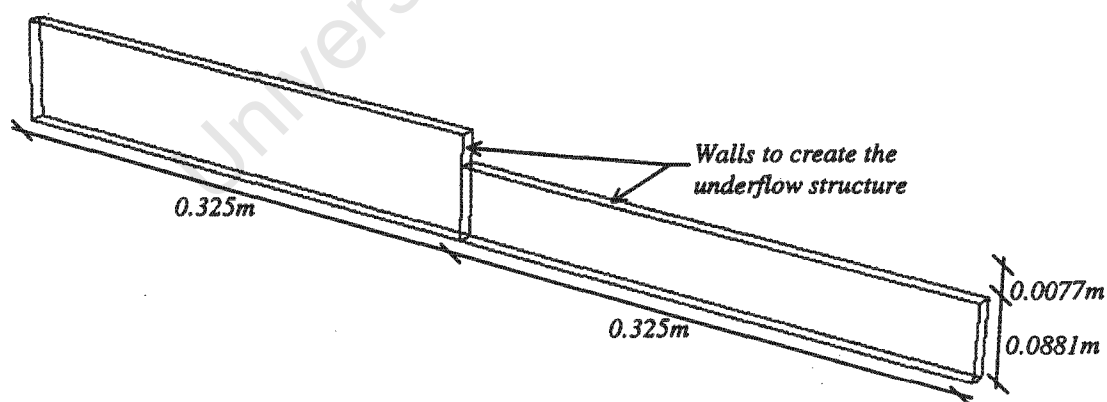


Figure 7.6 : Typical dimensions of the underflow structure geometry

The channel section is simulated from 0.325 m (physical model dimension) upstream of the underflow structure. The channel width is reduced to 10 mm, with symmetry conditions specified at the side boundaries, as the flow field is modeled two-

dimensionally. The model flow rate was 0.3271 l/s and the measured laboratory water depth was 95.8 mm. The surface was modeled as a moving wall parallel to the level channel bed. The flow patterns around the underflow structure for model B8 are shown in figure 7.7

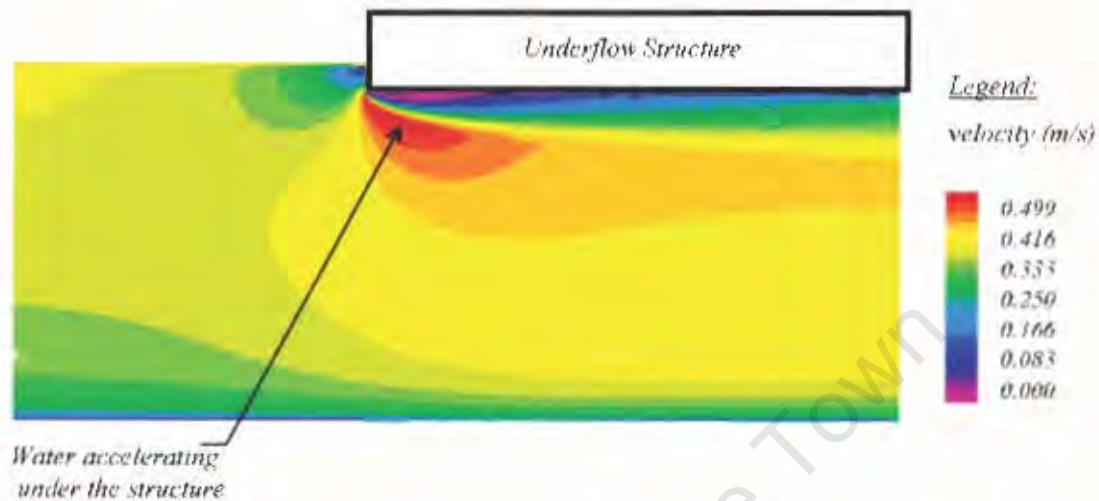


Figure 7.7 : Flow patterns around the underflow structure

The flow direction is from left to right, with the maximum velocities occurring in the red region below the underflow structure. In the upper area prior to the structure, the flow is slowed to zero, causing a steep velocity gradient from the surface to the depth of the structure below the surface. The velocity vectors are depicted in figure 7.8.

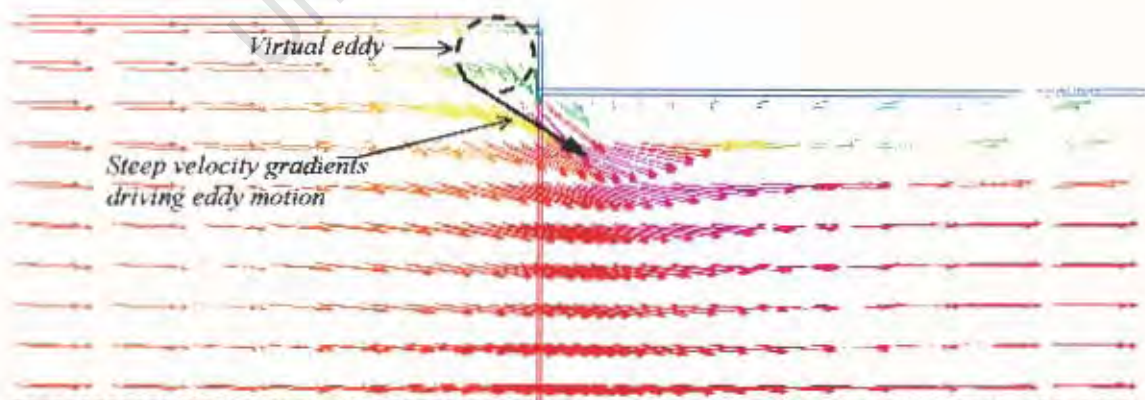


Figure 7.8 : Vector plot of velocities in the vicinity of the underflow structure

A virtual eddy extends approximately to the depth of the underflow structure below the water surface. The velocity distributions across the centre of this virtual eddy are shown in figure 7.9.

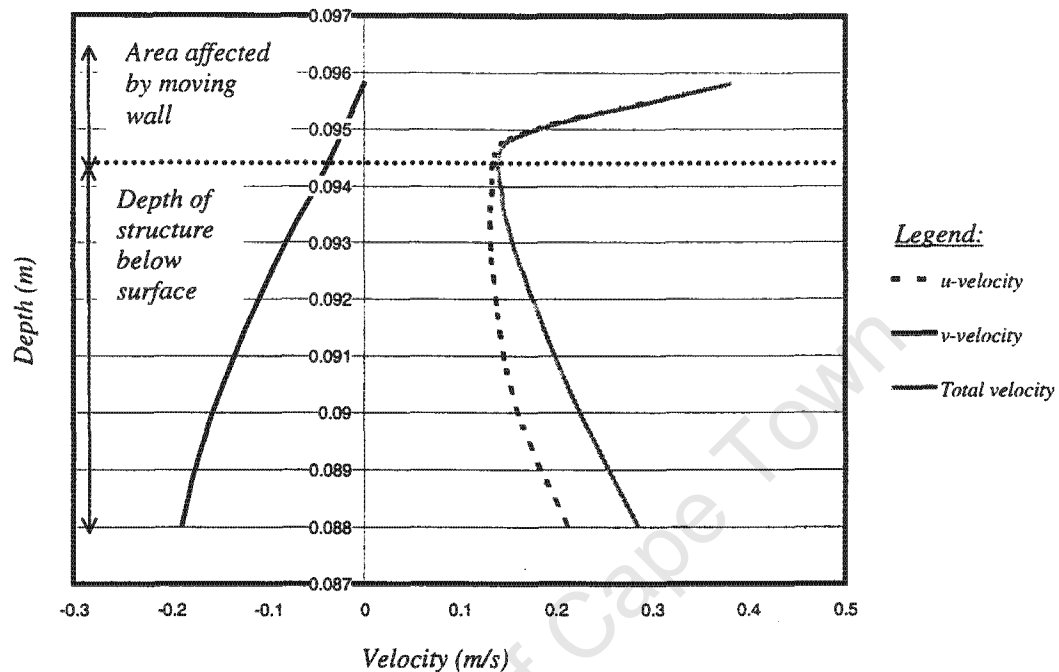


Figure 7.9: Velocity distributions with depth

The total velocity is nearly linear until the influence of the moving wall at the free surface is felt. The vertical or 'v-velocity' vectors are negative as flow tends to move down and under the structure with a velocity that increases with depth. This results in a large total velocity that drives the eddying motion. This total velocity distribution is characteristic of a forced vortex (Section 2.5) and the resulting velocity gradients cause increased power dissipation prior to the structure.

The stream power distribution was generated (fig 7.10) using the power dissipation theory of Section 3.4.6. 100 % of the cork beads were trapped. The maximum stream power zone has therefore been set at $\geq 1090 \text{ W/m}^3$ i.e. the power requirement of these beads.

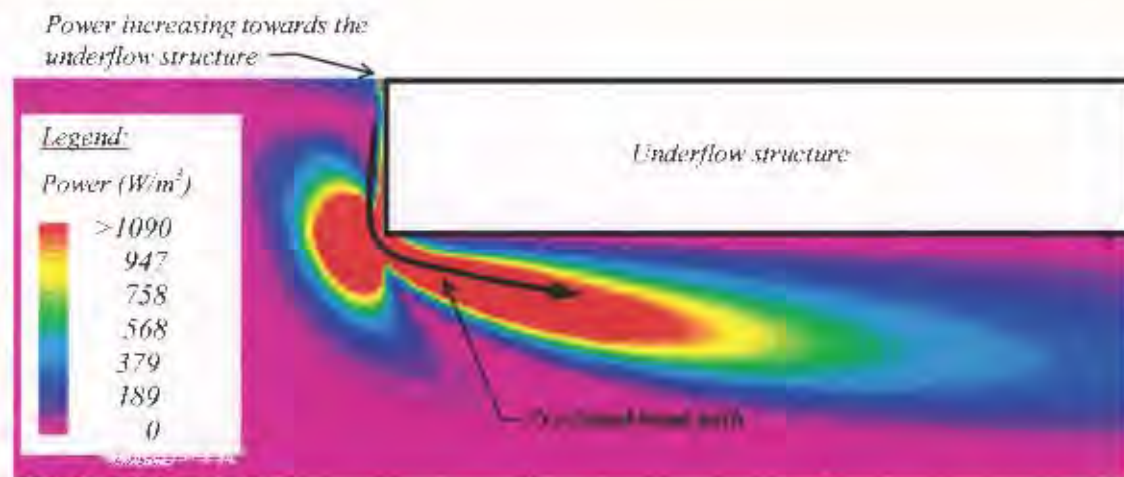


Figure 7.10 : Unit stream power distribution in the vicinity of the underflow structure

The unit stream power distribution indicates a red region of high power under the structure and a narrow band of higher stream power that stretches upstream of the structure towards the water surface. There is a small red region at the water surface indicating an increased power dissipation just prior to the structure. The predicted bead path is shown. The bead will be drawn below the surface due to the increasing power as the flow approaches the structure. The bead will then be drawn further down, below the underflow structure, in the turbulence associated with the high powers below. A maximum unit stream power of 4580 W/m^2 is ultimately reached.

The maximum power values below the underflow structure and the power values attained at the surface are summarised in table 7.6 for the remaining nine underflow structures that were numerically modeled. These are compared to the power requirement of the largest beads to pass under the underflow structure.

The power values at the surface predict the bead movement accurately for $\eta = 1$, i.e. $P_t = P_r$. Where the results deviate from this, the available power P_t is similar to the required power value P_r , i.e. the critical condition for suspension.

Model no.-	Maximum surface power (W/m ³)	Maximum power under the underflow structure (W/m ³)	Power requirement of largest bead to pass under the underflow structure (W/m ³)
B1	1340	3380	1090
B2	860	4440	>> 2960 required
B3	*	*	1090
B4	1090	9970	>> 2960 required
B5	970	10650	1090
B6	2850	13270	2960
B7	1270	13160	1090
B8	1240	4580	1090
B9	612	4580	>> 2960 required
B10	179	980	>> 2960 required

* Erroneous

Table 7.6 : Maximum power values at the surface and below the structure

7.3 Movement through a Contraction and over a Weir

Experiments were undertaken (Feuilolley 1999) to observe the movement of beads through a contraction and over a weir. The level flume had an initial width of 610 mm and this was abruptly contracted to a width of 320 mm. A weir was placed downstream of the contraction. Feuilolley released beads in the upstream section and observed whether they were trapped by the weir or passed over it. He then investigated Compion's (1997) criterion for predicting the bead movement. Rooseboom's (1992) rigid body analogy was extended to include a horizontal eddy and a vertical eddy to determine the total stream power. The horizontal eddy was positioned in the vena contracta and the vertical eddy was positioned prior to the weir (fig 7.11). The power requirement to suspend a bead could be compared to the stream power resulting from the combination of the contraction and the weir.

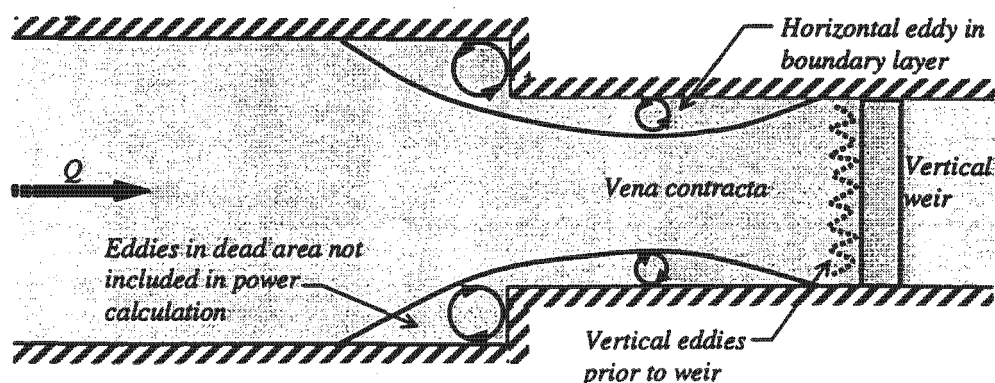


Figure 7.11 : Vortex development around a contraction (Feuillolley 1999)

The bead densities, settling velocities and individual power requirements are summarised in table 7.7.

Bead description:	V_u (mm/s)	ρ_b (kg/m ³)	P_r (W/m ³)
Green I	75	1060	40
Green II	124	1120	140
Brown	149	1170	247
White	187	1190	340

Table 7.7 : Description of Feuillolley's (1999) beads

Six different configurations were experimented with by Feuillolley (1997) and these were all modeled numerically. The flow details for the experiments and the corresponding numerical models are summarised in table 7.8.

The results for one contraction are presented here (model C1, table 7.8) and the results for the remaining five are presented in the graphs and tables. Figure 7.12 illustrates the contraction geometry, which was modeled symmetrically. The model flow rate was 1.2 l/s as it was modeled symmetrically. The surface is modeled as a horizontal moving wall, with the velocity varying according to equation 6.2 (Section 6.3.1) where the maximum velocity is at the symmetry plane and the minimum velocity is at the side wall boundaries.

Model no.	Exp. no.	Flow rate (l/s)	Initial width (mm)	Final width (mm)	Weir height (mm)	Water depth (mm)	Particles passing over weir			
							green I	green II	brown	white
							%	%	%	%
C1	1(1)	2.4	610	320	31.4	56	100	10	5	0
C2	2(1)	3.5	610	320	31.4	62	100	45	50	15
C3	3(1)	4.6	610	320	31.4	67	100	90	85	0
C4	1(2)	2.2	610	320	32.7	54	55	20	15	0
C5	2(2)	3.3	610	320	32.7	61	100	75	70	10
C6	3(2)	4.5	610	320	32.7	66	100	90	80	0

Table 7.8 : Experiment and model data for the six contractions

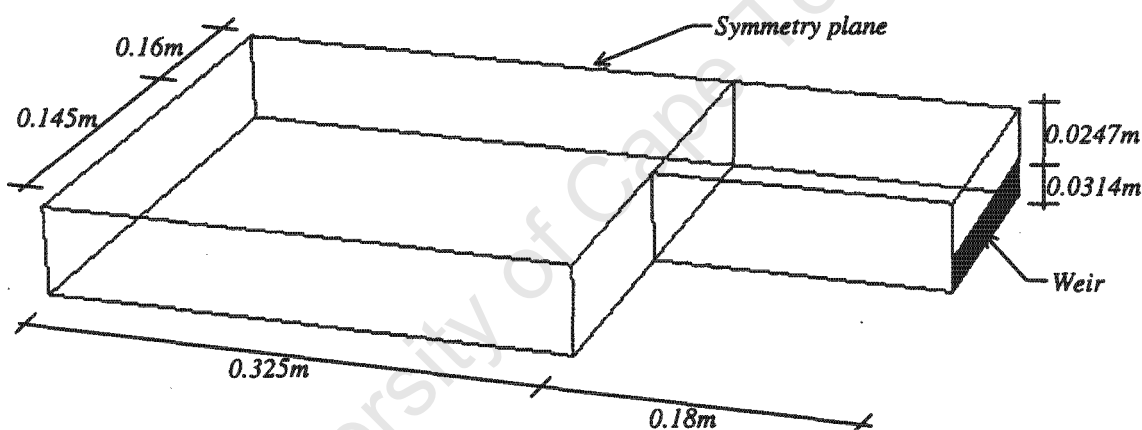


Figure 7.12 : Typical dimensions of the symmetrical contraction geometry

The CFD velocity magnitudes for the flow pattern through the contraction near the bed ($y = 0.01$ m) and at the weir height ($y = 0.0314$ m above the bed) are shown in figure 7.13.

The flow direction is from left to right. The red and pink regions depict the maximum and minimum velocities respectively. The flow accelerates in the contraction due to the reduced cross-sectional area. The flow also accelerates over the weir, resulting in a steep velocity gradient between the channel bed and the water surface. This provides an additional power input to the power dissipation that occurs at the sudden

contraction. This power is then dissipated into smaller eddies as the flow moves downstream.

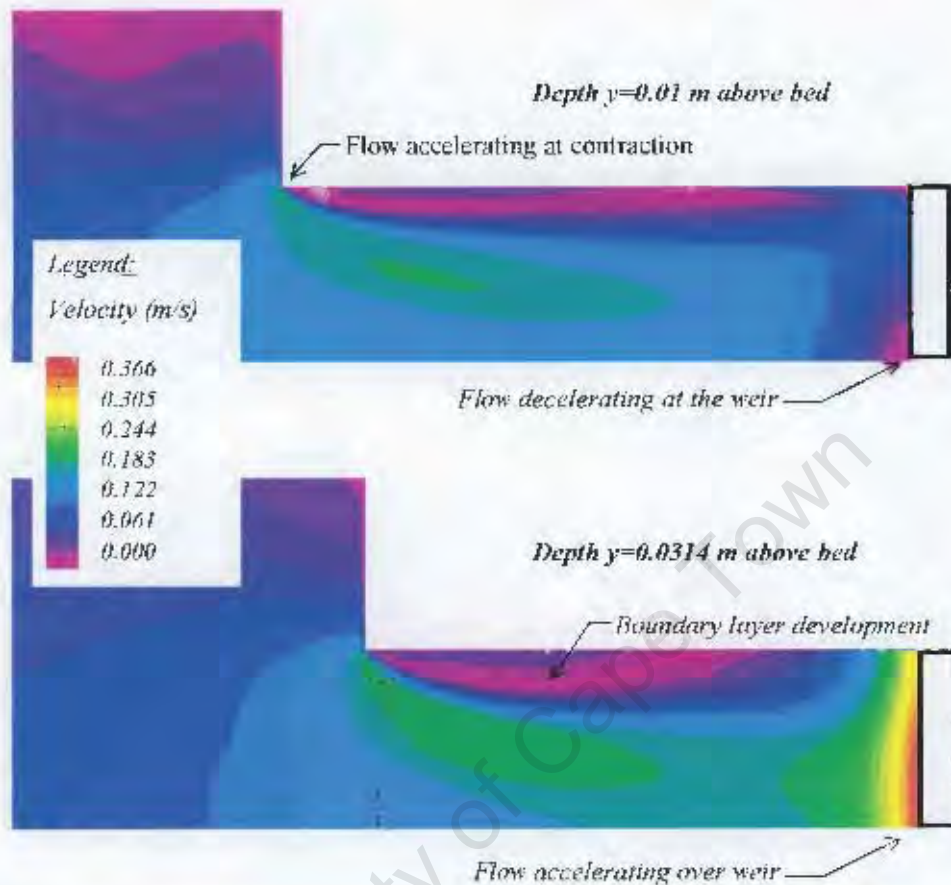


Figure 7.13 : Velocity magnitude distribution at the channel bed and at the height of the weir for the contraction

The velocity vector plot for the weir height is shown in figure 7.14. The flow field is more complex in the case of a three-dimensional numerical model compared with the two-dimensional models. In plan section, the horizontal eddy radii in the contraction boundary layer (see fig. 7.14) vary with depth from the largest at the surface to the smallest at the channel bed. This conical eddy distribution is further distorted as eddies occurring at greater depths are situated successively further downstream. Eddies therefore appear to originate at the channel bed and migrate further downstream as they increase in size. The velocity distribution across the eddy situated in the boundary layer at weir height is shown in figure 7.15.

The total velocity has a linear distribution that is at a maximum at the eddy circumference and a minimum at the centre, typically characteristic of a forced vortex. Both the u - and v -velocity components vary across the eddy with a linear velocity distribution.

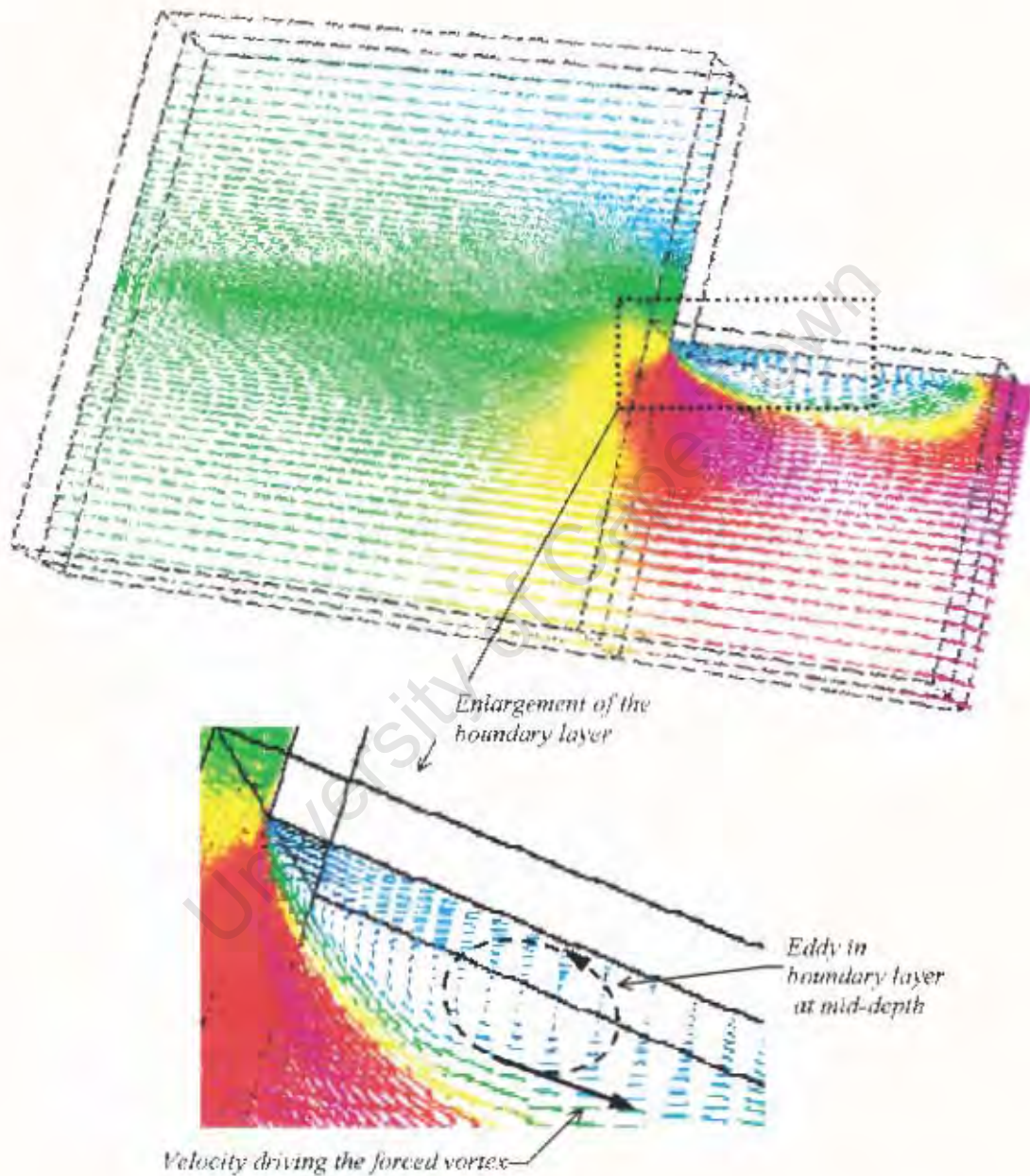


Figure 7.14 : Flow patterns in the vicinity of the contraction

The power distribution was generated for the various contractions. Figure 7.16 shows the power distribution at the channel bed and the weir depth for model C1. As the

Green II beads were the only bead type that all passed over the weir, the maximum power zone has been set at $\geq 40 \text{ W/m}^3$. At the channel bed, the power distribution suggests high power values in the boundary layer and at the vena contracta. At the weir depth, the highest power occurs at the weir. The red region represents the largest unit stream power magnitude.

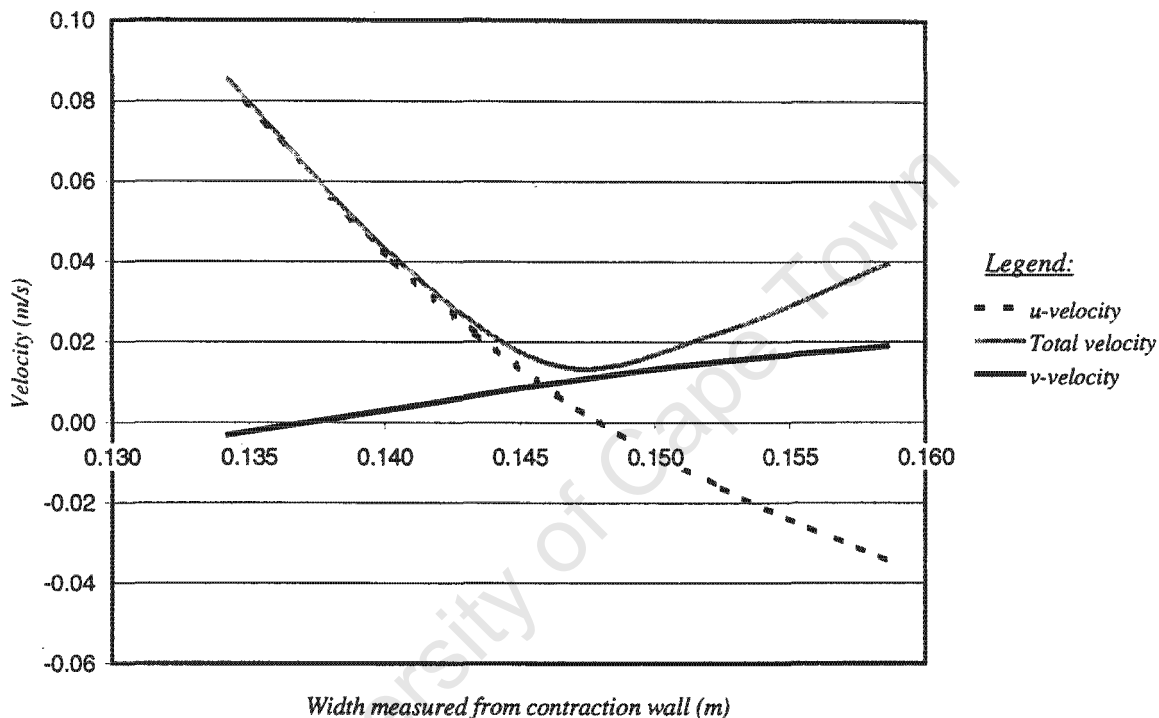


Figure 7.15 : Velocity distributions across the boundary layer eddy

The maximum values highlight the contribution from the velocity gradients to the power term. The flow velocity reduces to zero when it is in contact with the boundaries, and the adjacent water has extremely high velocities due to the geometric constrictions of the weir and the contraction, thus steep velocity gradients occur. These steep gradients result in large shear stresses and stream power dissipation.

The bead is suspended in the water as a result of the large power dissipation at the vena contracta. This suspension is maintained through additional power that is dissipated in the boundary layer and at the weir. Once suspended, it then moves along a streamline which passes over the weir.

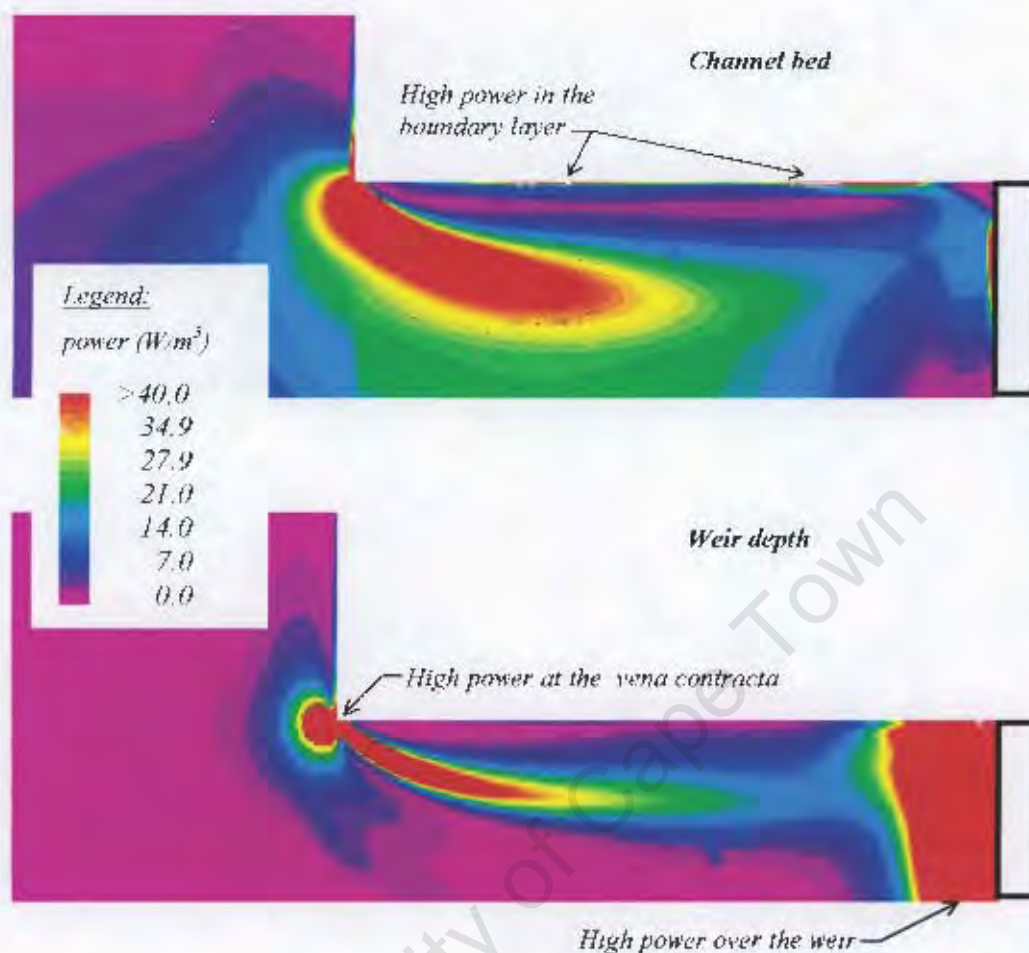


Figure 7.16 : Unit stream power distribution at the channel bed and at the weir height for the contraction

Table 7.9 summarises the maximum power values at the vena contracta and over the weir for the six contractions. These are compared to the power requirements of the largest beads that pass through the contraction and over the weir.

The power requirements of the largest beads passing over the weir correspond to the maximum value of the power dissipation at the weir. The maximum power at the vena contracta is generally lower than that required to suspend a particle.

Model no.	Maximum power at vena contracta (W/m ³)	Maximum power over weir (W/m ³)	Power requirement of largest bead to pass over the weir (W/m ³)
C1	126	258	247
C2	264	395	340
C3	363	464	247
C4	67	230	247
C5	255	341	340
C6	438	236	247

Table 7.9 : Maximum power at vena contracta and over the weir

7.4 Movement through an Expansion and over a Weir

Experiments were undertaken by Lyons (1999) to observe the movement of beads through a sudden expansion and over a weir. The experiments were conducted in a 610 mm wide, rectangular level flume. A 310 mm wide contraction was placed upstream and this was abruptly expanded into the 610 mm wide flume. The weir was placed further downstream of this expansion to trap the particles. The study was based on using the unit stream power approach to predict the likelihood of a bead passing or being trapped by the weir. Figure 7.17 shows the vortex streets that contributed to the stream power dissipation.

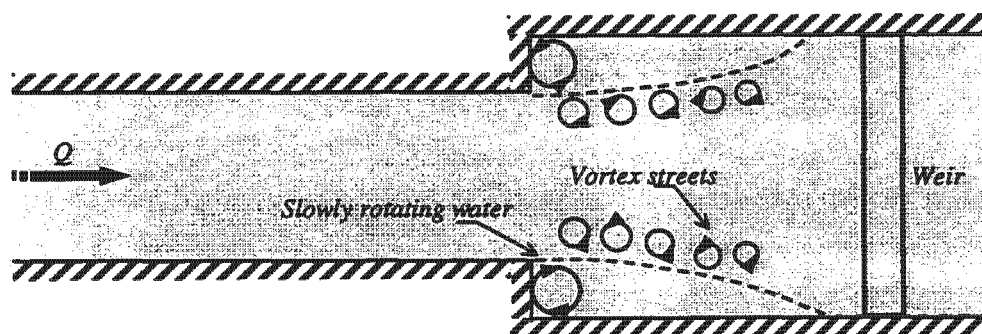


Figure 7.17 : Vortex development around an expansion (Lyons 1999)

Beads were introduced upstream of the expansion and the bead characteristics, as presented by Lyons (1999), are summarised in table 7.10.

Bead description:	V_{tr} (mm/s)	ρ_b (kg/m ³)	P_r (W/m ³)
Blue glass	650	2440	9130
Black	270	1210	561
White	151	1150	227
Navy blue	125	1080	103
Yellow	100	1100	85
Pink	87	1060	55
Navy blue (+)	74	1030	20

Table 7.10 : Description of Lyon's (1999) beads

One of these experiments (Lyons 1999) was modeled numerically. This experiment had a flow rate of 3.24 l/s, a measured depth of 55 mm and the downstream weir height was 31 mm. The weir trapped 100 % of the blue glass and black beads, while the remaining beads all passed over the weir. The model geometry, which is modeled symmetrically, is shown figure 7.18.

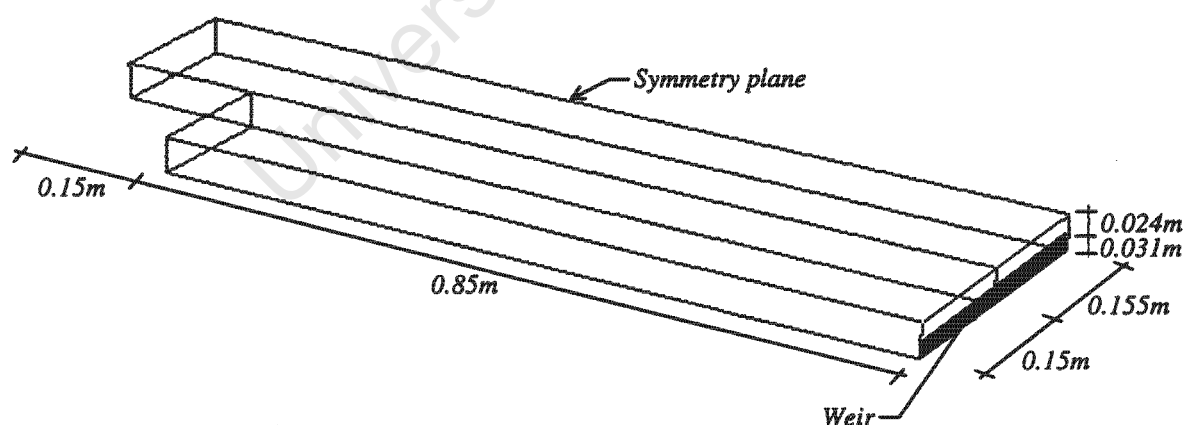


Figure 7.18 : Geometry of the expansion for the numerical model

The resulting velocity magnitude distribution at the expansion is shown in figure 7.19. The flow direction is from left to right. The water gradually reduces its speed as it

enters the expansion due to the increased area. An area of 'dead' water forms, where the flow slowly rotates, in the newly expanded corner. The flow then expands outwards over the distance prior to the weir, finally accelerating over the weir. The velocity vectors are shown in figure 7.20.

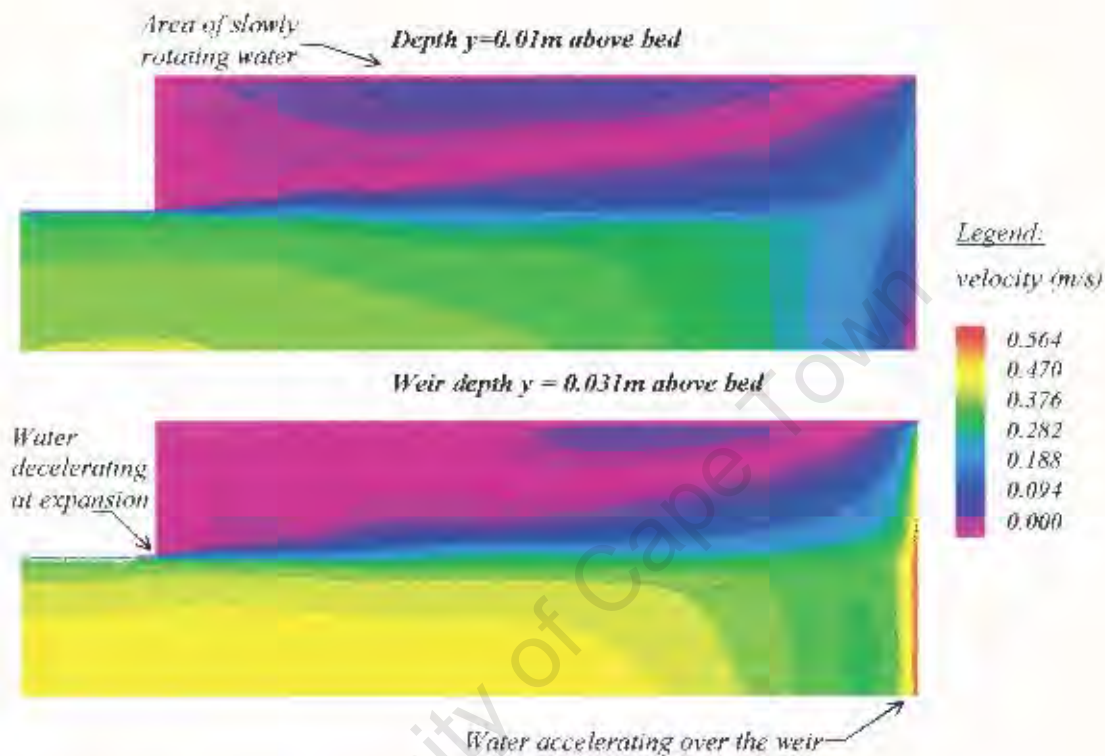


Figure 7.19 : Velocity distribution at the sudden expansion

The velocity slows down in the recirculation zone, however in the channel centre the velocity magnitudes are still high. This sets up a steep velocity gradient in the mixing layer, and a vortex street was observed in the laboratory. This vortex street is not present in the numerical model due to the surface moving-wall approximation, which forces the velocities at the surface to remain in the downstream direction. Furthermore, vortices are characterised by pressure drops towards the centre, and the moving wall dampens this effect. All other flow phenomena are, however, shown.

The stream power distribution was generated at weir height using the dissipation function (Section 3.4.6). Figure 7.21 shows the power distribution in the expansion region. As the navy blue beads were the beads with the largest power requirement

that traversed the weir, the maximum stream power zone has been set at $\geq 103 \text{ W/m}^3$, the power requirement for suspension of these beads.

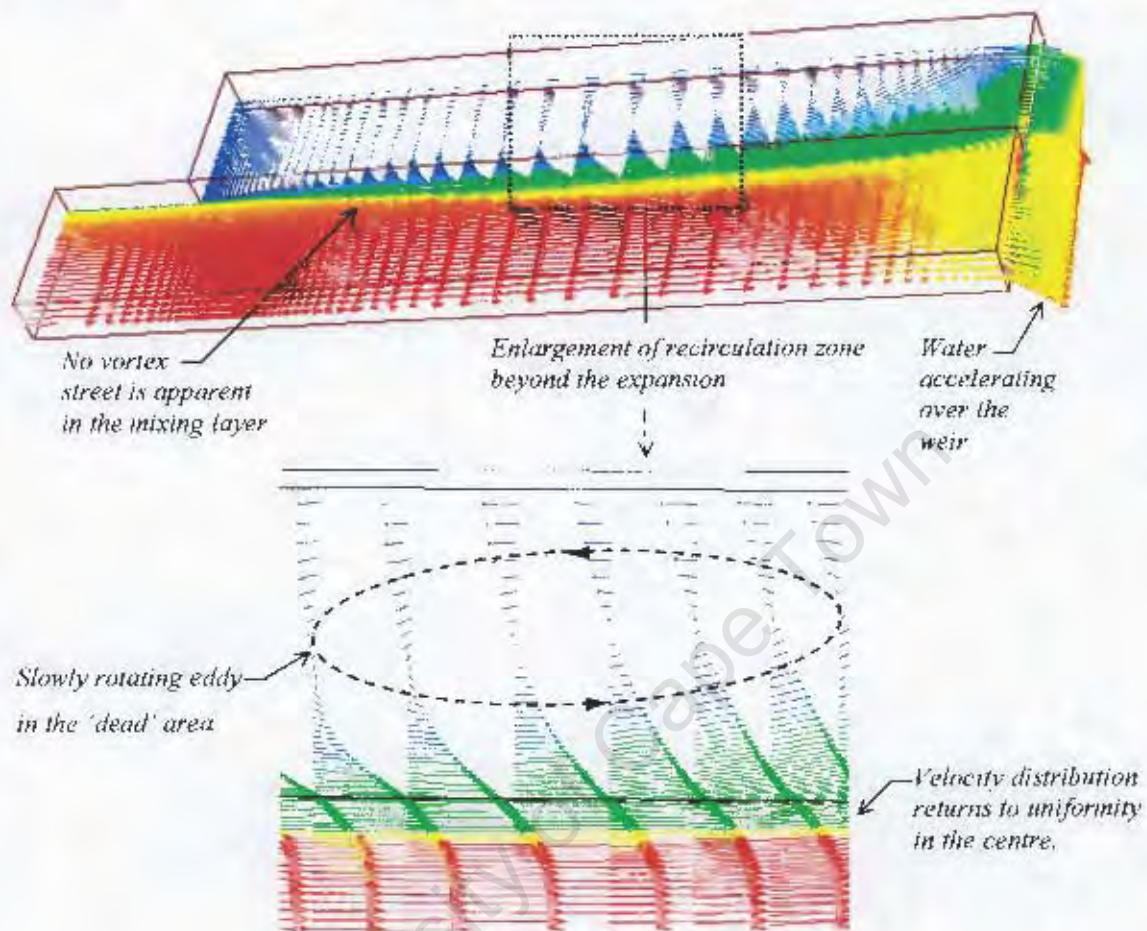


Figure 7.20 : The velocity vectors through the expansion

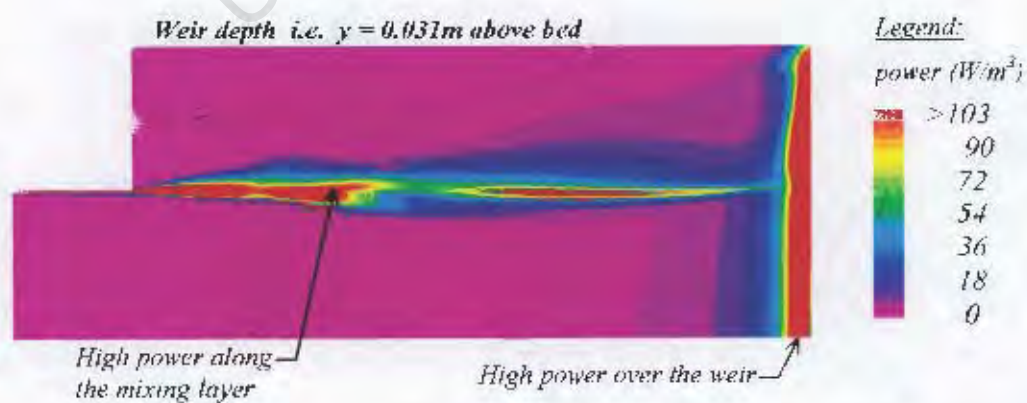


Figure 7.21 : Unit stream power distribution at the expansion and in the vicinity of the weir

The power distribution shows the region of high stream power in the mixing layer and at the weir. The particle is likely to be suspended at the expansion due to the high power at the separation point. The bead will then move along a streamline that moves through the mixing layer and over the weir. Additional power is dissipated in the mixing layer and adjacent to the weir to maintain the suspension of the particle.

7.5 Conclusions

The results for the prediction of bead movement around various hydraulic structures have been presented in this section. The flow pattern in the vicinity of the geometry changes illustrated the eddies contributing the most to the unit stream power. The velocity profiles across these eddies were in most instances characteristic of a forced vortex.

Unit stream power distributions were introduced for each structure. The power values were generally found to correlate with the bead path, although in some instances the values were lower than expected. This can be attributed to the stochastic variation in the unit stream power P_i about the mean $\bar{P}_i$ e.g. $P_i = \bar{P}_i + P'_i$ and if P'_i is positive then P_i is sufficient to suspend the bead. Once suspended the bead tends to follow the path of the streamlines in the water and thus be propelled past the relevant structure.

The power variation due to coherent flow structures in open channel flow is not well researched, yet it is essential in determining the total power dissipation. The limitation of the power distribution method for the three-dimensional suspension of beads, i.e. the sudden contraction and expansion, is the depth at which the distribution should be considered. For this study, the depth of the probable bead path was selected.

Chapter 8

Numerical Predictions of Scour and Deposition

This chapter gives a detailed description of the results obtained from the numerical prediction of sediment scour and deposition, and their comparison to the actual observed profiles. These profiles are predicted by implementing the unit stream power dissipation function (Section 3.4.6) in the numerical model, and hence generating the unit stream power distribution. This power distribution is compared to the sediment particle power requirements with an η value of $\frac{1}{67}$ for rough boundaries (Section 3.3.3), to determine the scour patterns. The distribution of the Movability Number is also plotted as an η value of $\frac{1}{67}$ corresponds to a Movability Number of 0.17 for rough boundaries. All the experimental cases that were modeled had rough boundaries.

8.1 Scour Profile Prior to a Vertical Weir

Harnett (1998) investigated the scour profiles upstream of a weir for a variety of sediment sizes. Uniform silica sand of three different diameters were separately placed onto the bottom of a 6.38 m long laboratory flume sloped at 1:127.6. The surface of the sand bed was made horizontal with a 50 mm high sharp-crested weir located at the downstream end of the flume. The bed was then subjected to a constant flow of clear water until a stable scour profile was achieved – generally after approximately 35 hours (fig 8.1).

One of these runs was selected and modeled numerically. The test case had a flow rate of 3.5 l/s over a sand with a median diameter of $d_{50} = 0.86$ mm and a settling velocity of 0.09 m/s. This was simulated numerically in both the initial and final condition (fig 8.1).

The power requirement P_r of an individual particle can be calculated from equation 3.8 (Section 3.3.2) as 1460 W/m^3 . The P_r value required for movement was therefore

21 W/m^3 for an η value of $\frac{1}{n^2}$ for incipient motion on a rough boundary (Section 3.3.3). The corresponding critical value of the Movability Number is 0.17.

The initial condition was modeled first to show the power values P_i at the bed were greater than 21 W/m^3 required for equilibrium conditions, i.e. the water had an erosive capacity. Similarly the values of the Movability Number were shown to be greater than 0.17 at the bed. To facilitate the interpretation of the plots, virtual Movability Numbers were generated for the whole of the water column using virtual shear velocities computed from equation 2.14 (Section 2.6) after the substitution of τ for τ_o . The actual Movability Number is therefore the bed value, which is the maximum value in the water column since the shear stress is a maximum at the bed. The virtual Movability Number decreases with the shear stress and thus decreases from the bed to the water surface.

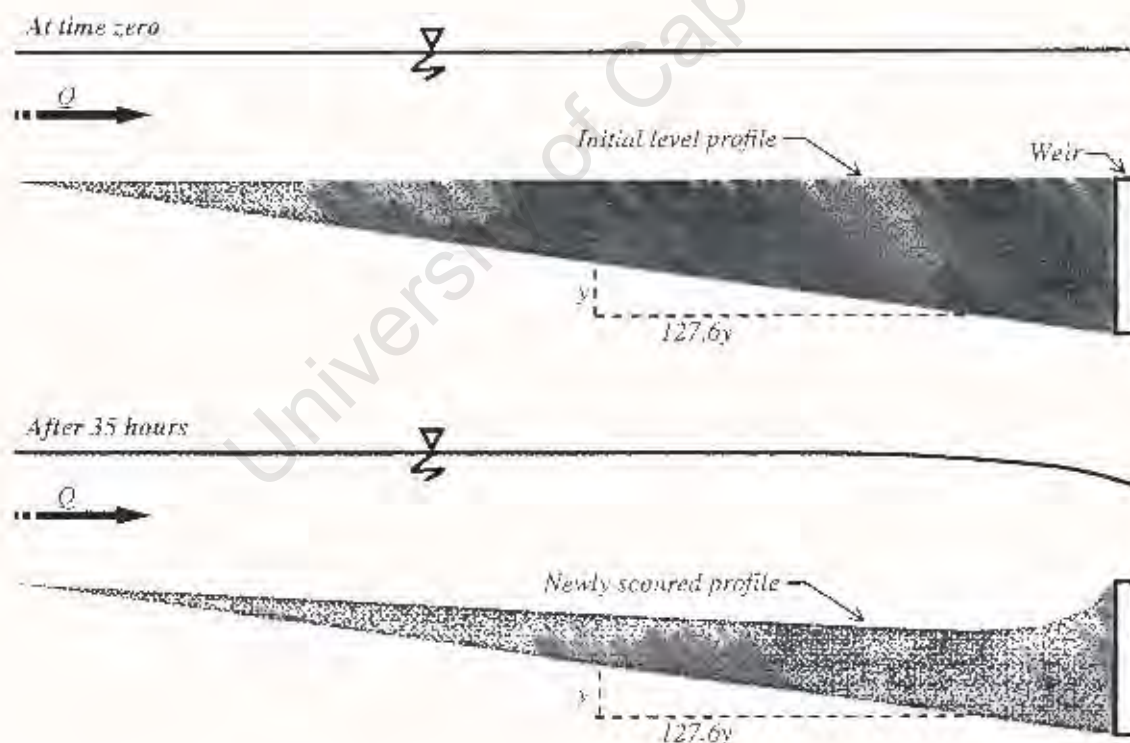


Figure 8.1 : The before and after experimental sediment profiles for the vertical weir (Harnett 1998)

Figure 8.2 shows the unit stream power distribution for the initial and final condition. The maximum unit stream power zone has been set at $\geq 21 \text{ W/m}^3$ for both plots and is represented by the red region. In the initial level analysis, a thin red line can be seen at the bed. The power increases rapidly at the bed due to the steep velocity gradients in this region, thus the red region is extremely narrow. In the final curved section, the red line is no-longer present, except immediately prior to the weir, thus there is in general no longer any potential for scour. This region where the scour potential is high but there is no apparent movement can be explained by noting that the particles are situated on an adverse slope, and are thus more resistant to movement. Armitage (2001) adapted the incipient motion criteria for topography (Section 3.5). However, this more complex approach was only implemented in the analysis of deep scour around a bridge pier (Section 8.4).

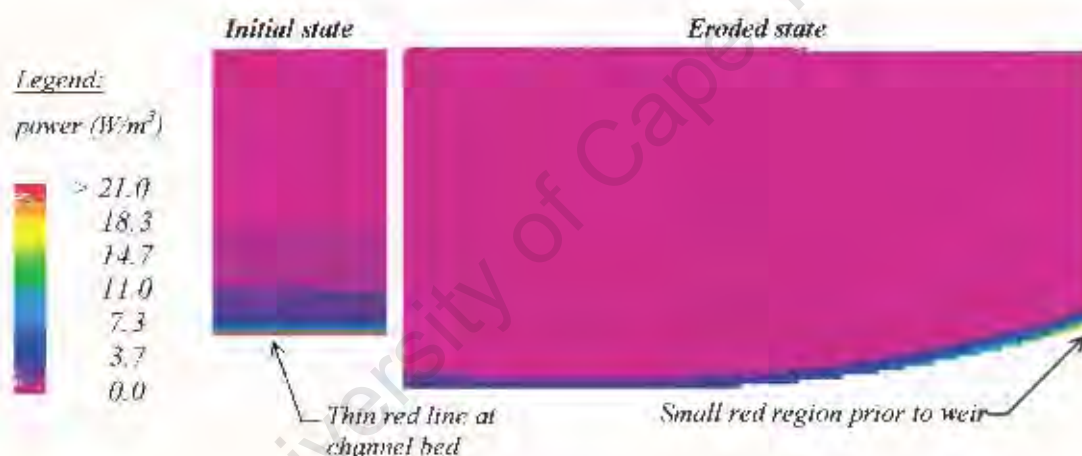


Figure 8.2 : Unit stream power distribution for the initial and final condition above the vertical weir

This situation is further confirmed by the plots of the Movability number (fig 8.3). The critical value for incipient motion is 0.17 (Section 3.3.3) and thus the maximum value in the plot has been set at ≥ 0.17 . In the analysis of the initial conditions, the value of the virtual Movability Number reaches 0.17 far above the channel bed, indicating a strong erosive capacity. In the analysis of the final profile, the value of 0.17 is only attained at the channel bed thus confirming that the scour profile is now approximately in equilibrium with the flow.

This unit stream power distribution supports the theory that it is possible to predict the scour pattern prior to a weir by considering either the unit stream power or alternatively the Movability Number at the bed.

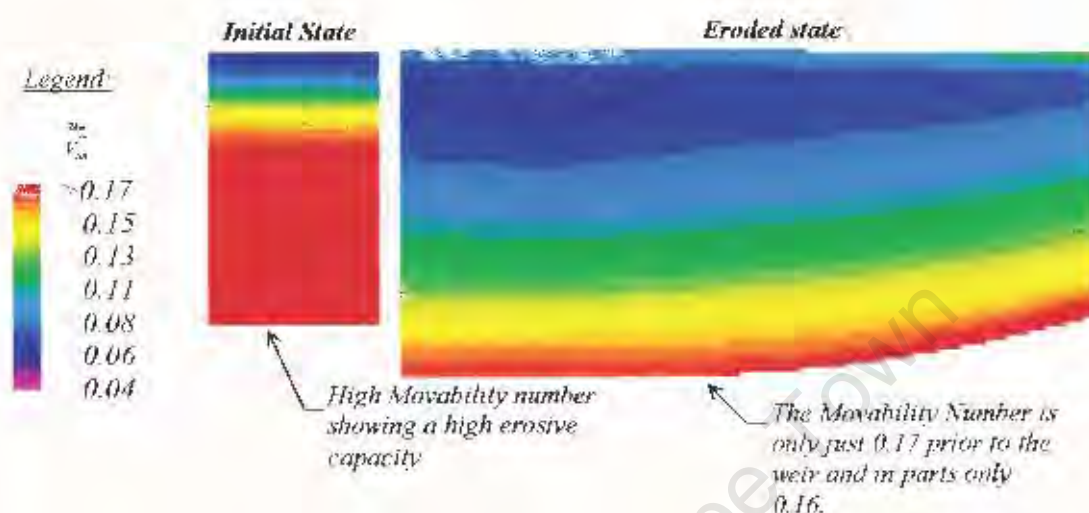


Figure 8.3 : The Movability Number for the initial and final states above the vertical weir

8.2 Scour at a Vertical-wall Abutment

Mitchell (2000) examined the scour of thin layers of uniform quartzitic sands of various sizes placed on a level floor around a vertical-wall abutment located in a 820 mm wide rectangular flume and subjected to a steady flow. The abutment had a length of 220 mm, was 35 mm thick, and was set at an angle of 90 degrees to the flow direction. Figure 8.4 shows the experimental layout. Approximately 1 litre of sediment was sprinkled evenly over the glass flume bed. The flow was then allowed to scour the material around the structure until equilibrium was reached after about 3 hours.

The test case involving a flow of 10.2 l/s, depth 100 mm, over a sand with a median diameter of $d_{50} = 0.55$ mm and a measured settling velocity of 0.089 m/s was

simulated by the numerical model. This settling velocity can be calculated based on the formulae presented in Section 3.1.5 and this yields settling velocities of 0.109 m/s and 0.067 m/s using equations 3.1 and 3.2 (Section 3.1.5) respectively. The mean of these two values was 0.088 m/s. As this was similar to the settling velocity measured in a settling velocity measuring column, the measured velocity of 0.089 m/s was used for the determination of the incipient motion power conditions and the Movability Number. The particle power requirement P_c can be determined from equation 3.8 (Section 3.3.2) as 1440 W/m^3 . The critical applied stream power for incipient motion could hence be calculated as approximately 21 W/m^3 for an η value of $\frac{1}{67}$ for incipient motion at a rough boundary (Section 3.3.3). Once again the critical Movability Number was in the order of 0.17.

Figure 8.5 shows the numerical model dimensions.



Figure 8.4 : Mitchell's (2000) experimental layout

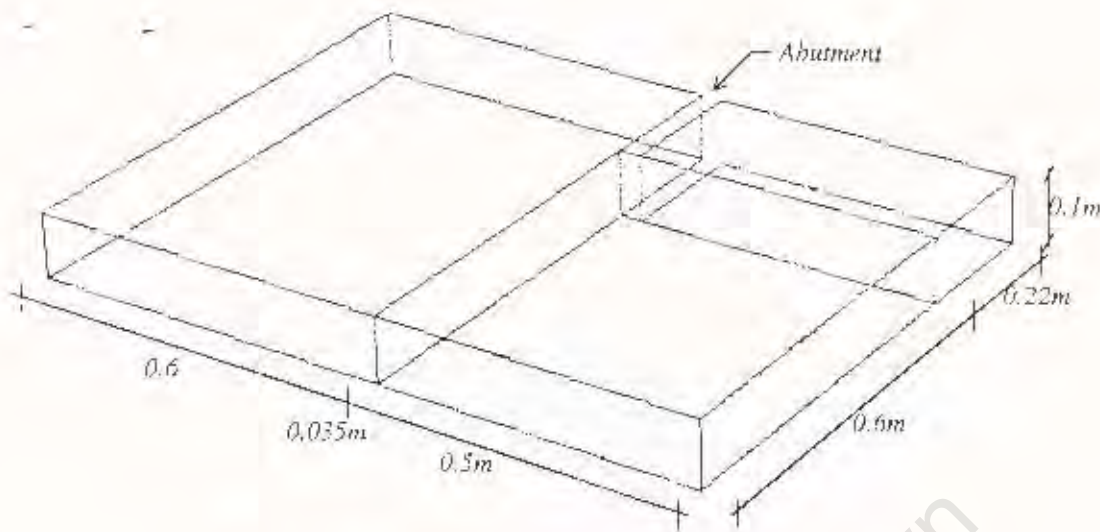


Figure 8.5 : The dimensioned model for the abutment

The velocity distribution that was generated around the abutment is shown at mid-depth in figure 8.6. In this abutment model there is evidence of a vortex street development. The eddy at the end of the abutment represents the starting eddy for the vortex street, however the movement of these eddies downstream is not identified due to the damping effect of the moving wall at the surface. The eddy in the recirculation zone is also clearly evident. The velocity distributions across these eddies have been plotted in figure 8.7 and 8.8 respectively.

The velocity plot in figure 8.7 shows the forced vortex that is present at the end of the abutment i.e. section A-A in figure 8.6. The flow then accelerates at a constant rate towards the centre of the channel. Similarly in figure 8.8, the forced vortex is apparent in the recirculation zone or 'dead' region (figure 8.6, section B-B), and then the flow accelerates out towards the channel centre. The flow is then slowed to a velocity of zero on the opposite bank.

The power distribution (fig 8.9) was generated for the abutment to determine the scour areas, based on the dissipation function (Section 3.4.6). The maximum power zone has been set at $\geq 21 \text{ W/m}^3$, which is the particle power requirement for incipient motion. Lines were plotted across these contour plots to indicate the limits of scour

In both cases the distribution is not identical to the actual scour profile. The power distribution reveals the shape of the scour more accurately, while the Movability Number is a far more conservative estimate of the region since the red area is larger than the observed scour pattern that is superimposed on it.

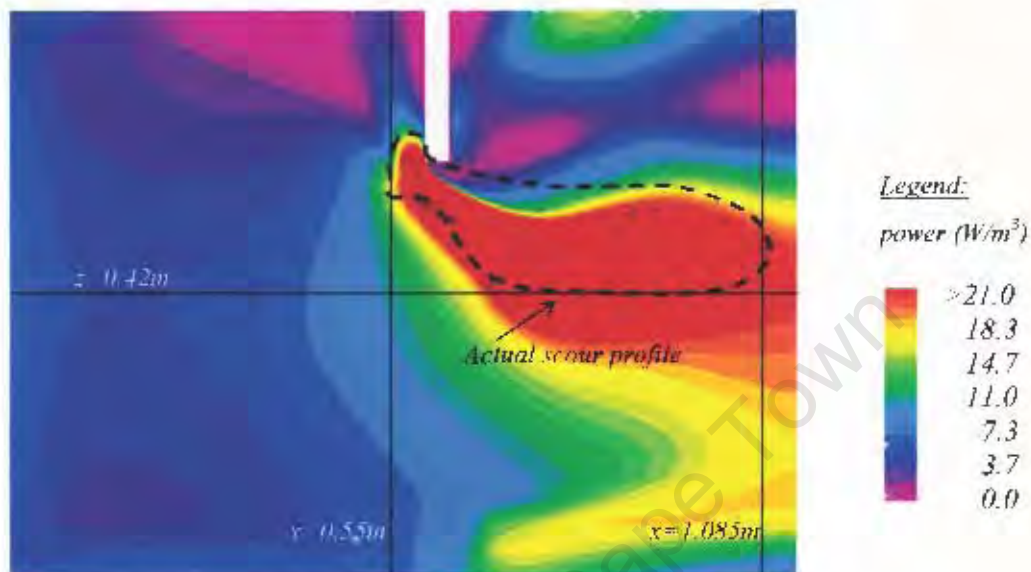


Figure 8.9 : Power distribution around the abutment

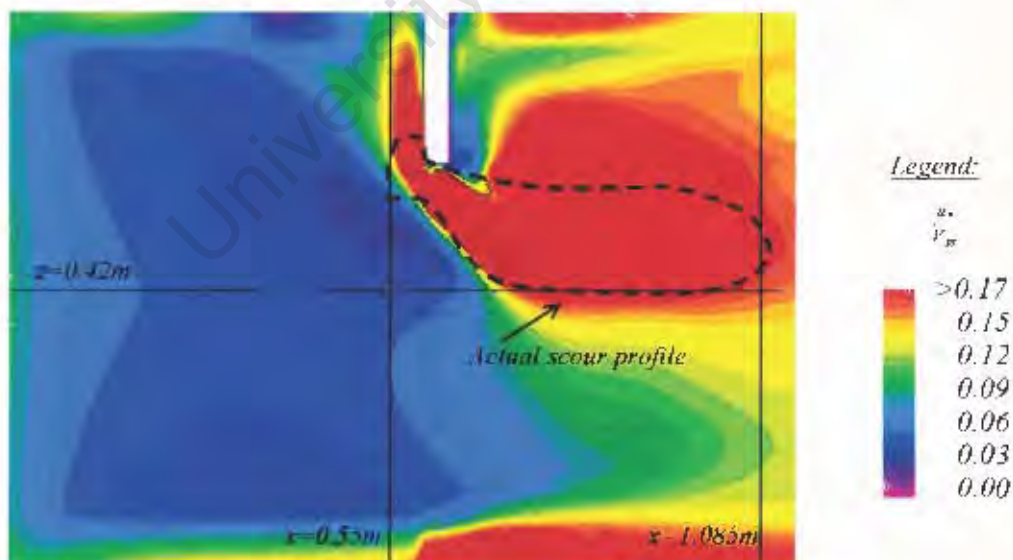


Figure 8.10 : Movability Number distribution around the abutment

8.3 Scour Patterns at a Cylindrical Pier

Midgley (2000) examined the scour of thin sediment layers of uniform quartzitic sands of various sizes placed around a 45 mm diameter circular pier centrally located in a 605 mm wide flume and subjected to a steady flow. Approximately 1 litre of sediment was sprinkled evenly over the flume bed. The flow was then allowed to scour the material around the structure until equilibrium was reached after about 3 hours.

The test case involving a flow of 7.69 l/s, depth 74 mm, over a sand with a median diameter of $d_{50} = 0.55$ mm and a measured settling velocity of 0.089 m/s was simulated by the numerical model. This settling velocity was calculated as before (Section 8.2) as 0.088 m/s. Once again, the particle power requirement P_p was determined from equation 3.8 (Section 3.3.2) as 1440 W/m^3 , and the critical applied stream power for incipient motion was 21 W/m^3 . The critical Movability Number was in the order of 0.17.

The dimensioned geometry, that was modelled symmetrically to reduce the number of cells used, is shown in figure 8.11.

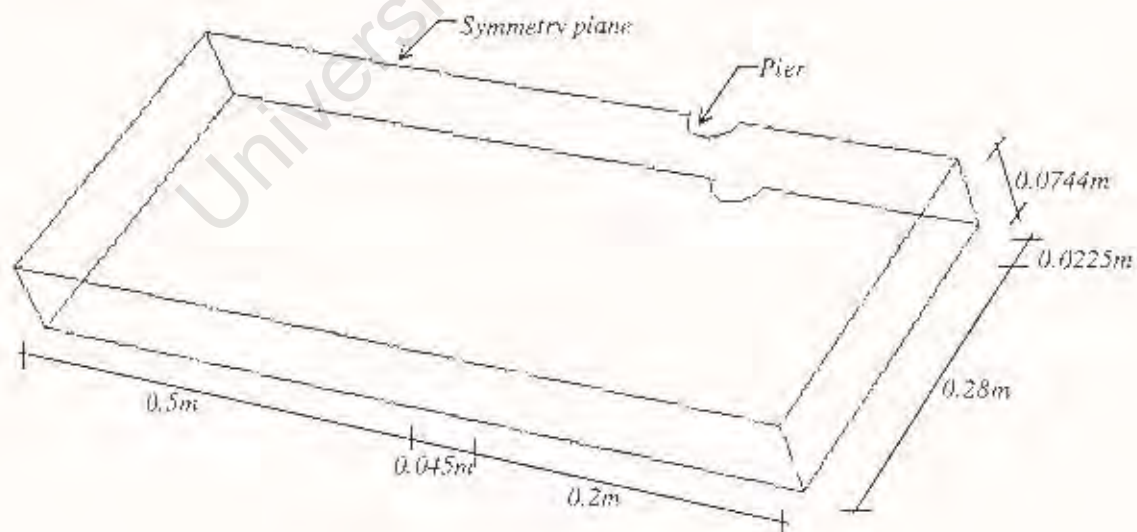


Figure 8.11 : The pier geometry

The flow pattern around the pier is illustrated with a velocity vector plot (fig 8.12).

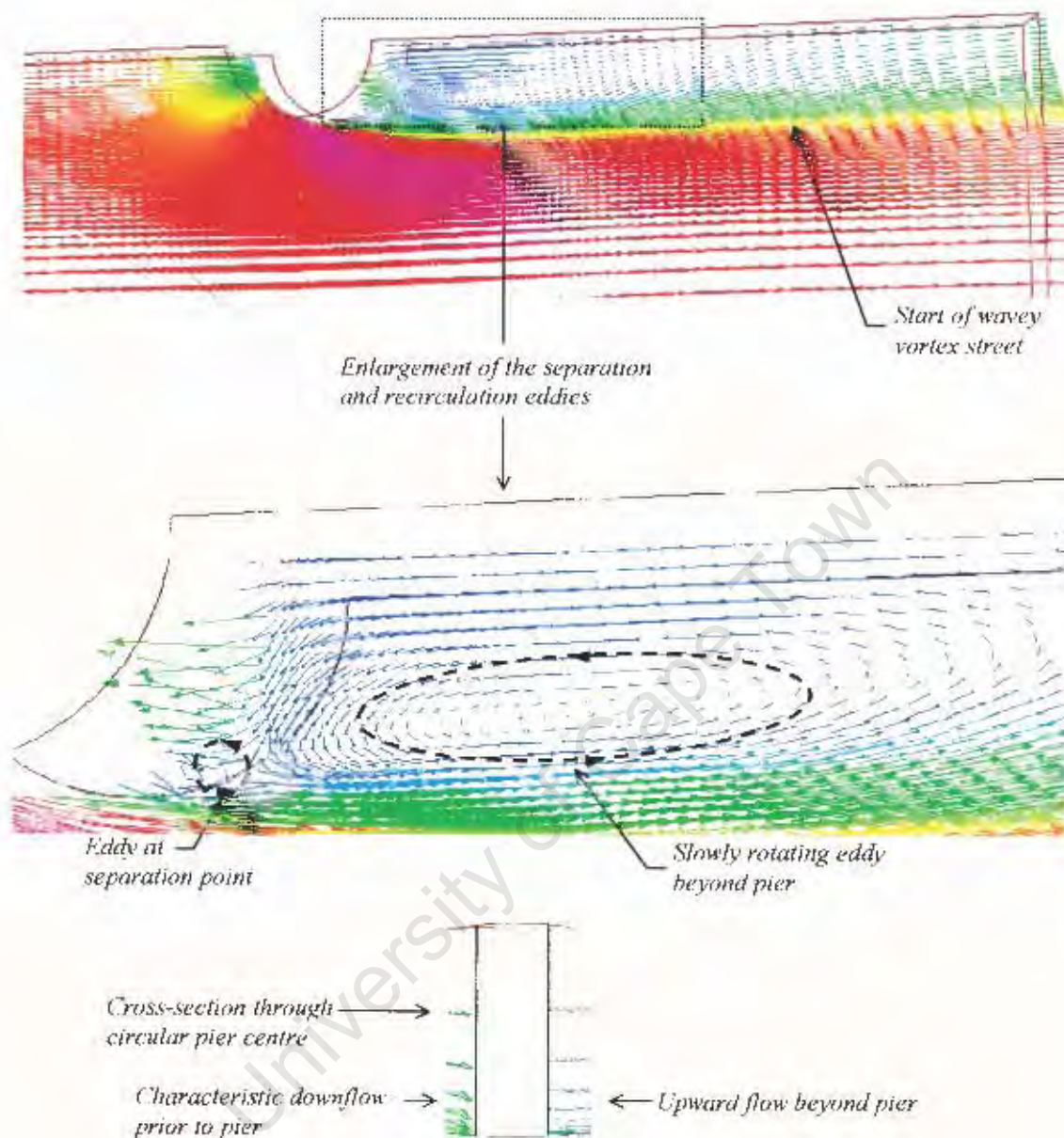


Figure 8.12 : Velocity vectors at the pier

Although the eddy at the separation point can be identified, wake vortices are not apparent. The horseshoe vortex is also not identifiable, as the mesh size is too coarse except immediately adjacent to the bed. Also the free-surface rise has been prevented. The large slowly rotating forced vortex beyond the pier is clearly present. Other flow characteristics such as the downflow prior to the pier and the up-draught beyond the pier can be seen in the cross-section.

The unit stream power distribution was also generated at the channel bed to facilitate scour prediction (fig 8.13). This distribution was plotted at a depth of 0.00055 m, as this is the average height of the sediment above the channel bed. The maximum stream power zone has been set at $\geq 21 \text{ W/m}^3$, as this represents the equivalent P_c value for incipient motion.

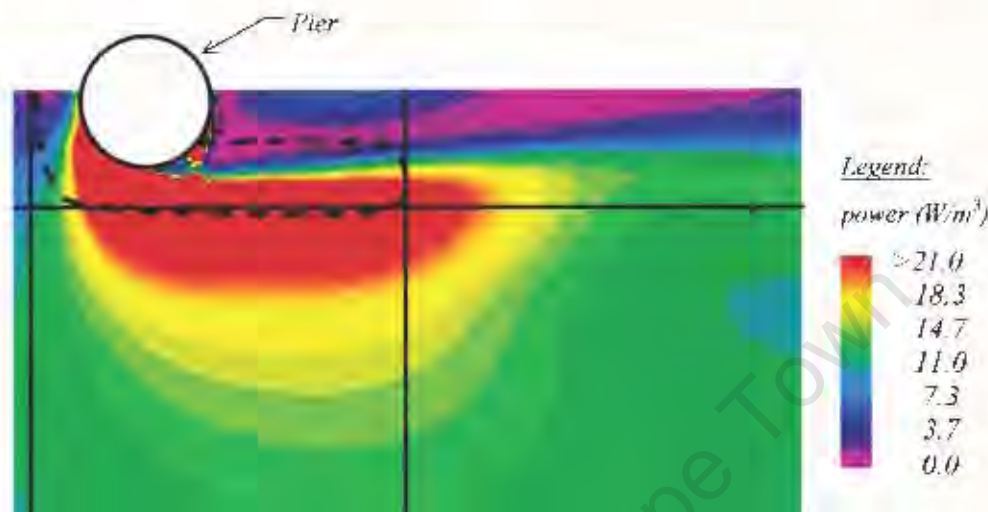


Figure 8.13 : Unit stream power distribution around the pier

The actual scour profile has been superimposed onto the predicted pattern of the model. The power plot reveals that although the shape of the scour profile is similar to the observed profile in the laboratory, its location has been shifted slightly away from the line of symmetry. This shift in position of the scour pattern can probably be attributed to the surface being modeled as a horizontal moving wall, which will tend to raise the pressure on the line of symmetry above what it would be in reality. In reality a slight drop in surface water elevation and sediment deposition characterises the downstream region. The predicted scour prior to the pier has a narrower band than the scour that actually occurs.

Figure 8.14 shows the Movability Number distribution at a position 0.00055 m above the channel bed, i.e. at the particle height. The maximum region has been set at the incipient motion value of ≥ 0.17 (Section 3.3.3). The Movability Number is found to under-predict the scour where it does occur and it over-predicts the scour in the adjacent regions.

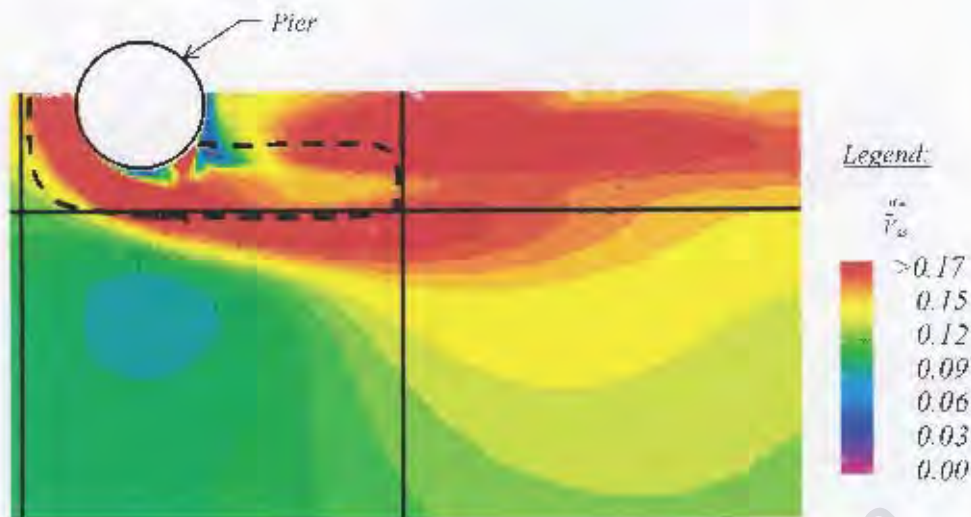


Figure 8.14 : Movability Number distribution around the pier

Of these two approaches the unit stream power distribution predicts the scour patterns more accurately.

8.4 Scour Hole Profiles at a Cylindrical Pier

Babaeyan-Koopaei (1996) investigated scour profiles around piers and abutments. He proposed a mathematical model to predict the temporal variation of scour depth in clear-water and live-bed scour conditions. Laboratory experiments were undertaken to measure the scour profiles around the pier or pier groups. A unique feature of this research was that the experiments were carried out in a regime flume, i.e. a self-formed laboratory channel (fig 8.15), resulting in the mobility of the entire cross-section. These represented natural field conditions where channels are generally non-rectangular, with mobile banks and possible overbank flow. The flume was 22 m long, 2.5 m wide flume with a depth of 600 mm. The particle mean diameter, d_{50} , was 1 mm.

One of these experiments, test no. 38B (Babaeyan-Koopaei 1996) was modelled numerically, in both the initial and final conditions.



Figure 8.15 : A typical self-formed channel (Babaeyan-Koopaei 1996)



Figure 8.16 : A typical channel with piers (Babaeyan-Koopaei 1996)

This experiment investigated the scour profile around a singular cylindrical pier, centrally located, with a diameter of 48 mm. The flow rate was 2.5 l/s and the average water depth was 129.5 mm. The maximum scour depth adjacent to the pier was 55.6 mm. Two conditions were modelled:

- (i) Prior to scour i.e. the self-formed channel (fig 8.15) with the pier installed.
- (ii) After development of the equilibrium scour profile around the pier e.g. fig 8.16 of group piles.

The numerical model was used to indicate the erosive potential of the channel in the vicinity of the pier for case (i) and to show that incipient motion conditions were present for case (ii) i.e. approximately zero scour potential at the channel bed.

The particle settling velocity was not measured experimentally. The settling velocity was therefore calculated based on the formulae presented in Section 3.1.5 and this yielded settling velocities of 0.147 m/s and 0.111 m/s using equations 3.1 and 3.2 (Section 3.1.5) respectively. An average settling velocity was determined by providing equal weighting to these computed values, giving an average settling velocity of 0.129 m/s. This settling velocity was implemented for determination of the Movability Number and other related equations throughout this section.

The model geometry for case (i) is shown in figure 8.17. A three-dimensional user patch was created adjacent to the bed as discussed in Section 6.11, and this is depicted in the vicinity of the pier (fig 8.17).

The incipient motion critical function (equation 3.48, Section 3.5) for sediment on an uneven bed was solved for at a depth of 0.5 mm above the channel bed. As the geometry for the initial condition, case (i), was created with a level bed, this critical function should be 0.17 over all of the channel bed i.e. the value of the Movability Number for incipient motion at a rough boundary (Section 3.3.3). These values are graphed for each cell centre at the boundary in figure 8.18. The values that are not

equal to 0.17 can be explained by the cells situated adjacent to the pier, where the velocity vectors are not parallel to the bed.

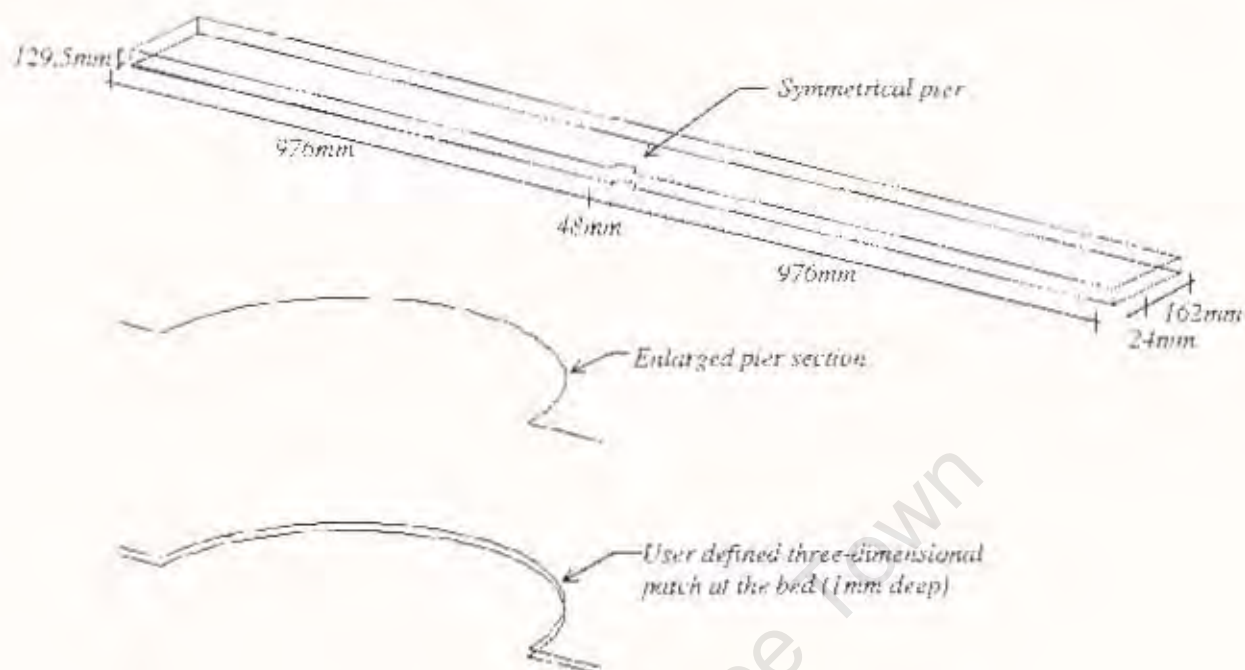


Figure 8.17 : The model geometry for the initial pre-scour condition

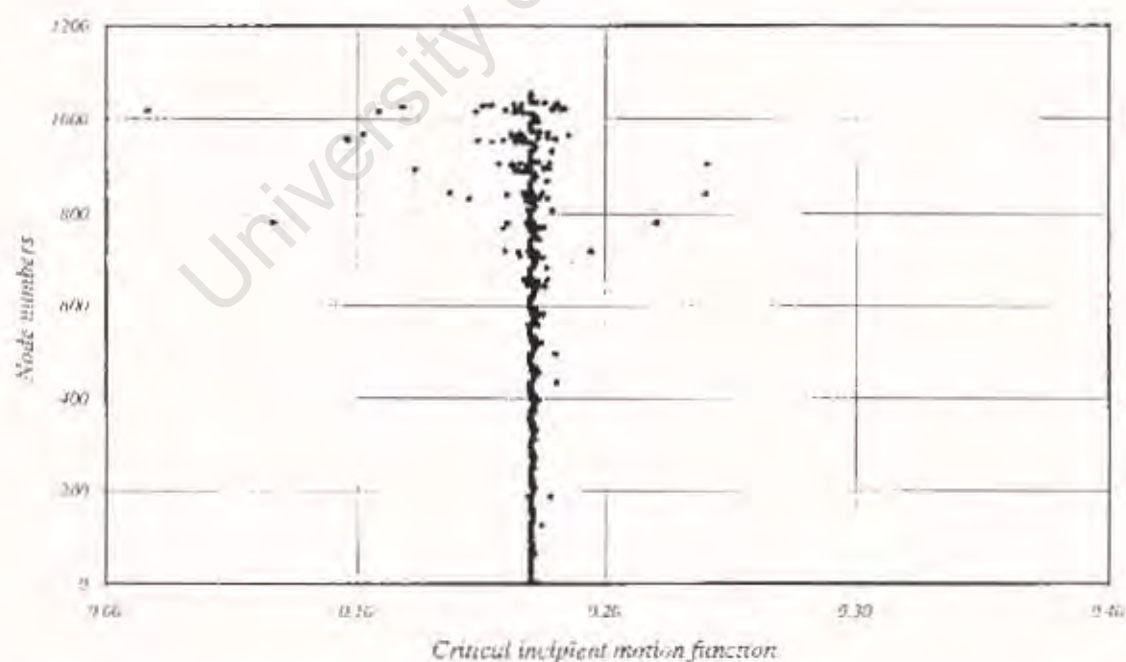


Figure 8.18 : The critical incipient motion function for each geometry nodal value

The Movability Number was solved for at a depth of 0.5 mm above the channel bed.
The critical incipient motion function

$$\left. \frac{u_*}{V_{ss}} \right|_{\beta, \gamma} = \psi \left. \frac{u_*}{V_{ss}} \right|_0$$

3.48

was then subtracted from this to give the scour potential,

$$\Omega = \left. \frac{u_*}{V_{ss}} \right|_{\Lambda_{actual}} - \psi \left. \frac{u_*}{V_{ss}} \right|_0$$

6.6



Figure 8.19 : The Movability Number distribution in the vicinity of the pier at a depth of 0.5 mm for the initial condition

The results were plotted in a spatial distribution using IDL (2000). Figure 8.19 shows the distribution of the Movability Number at the bed. The values are in the range of 0.2 to 0.4, which are larger than 0.17, indicating the erosive potential at the bed.

Figure 8.20 shows the distribution of the scour potential Ω (equation 6.6) where these values are positive and most data falls in the range 0.1 to 0.25, i.e. greater than zero, indicating the scour potential in the vicinity of the bed. The values increase dramatically adjacent to the pier, indicating the larger scour potential where the scour hole is anticipated.

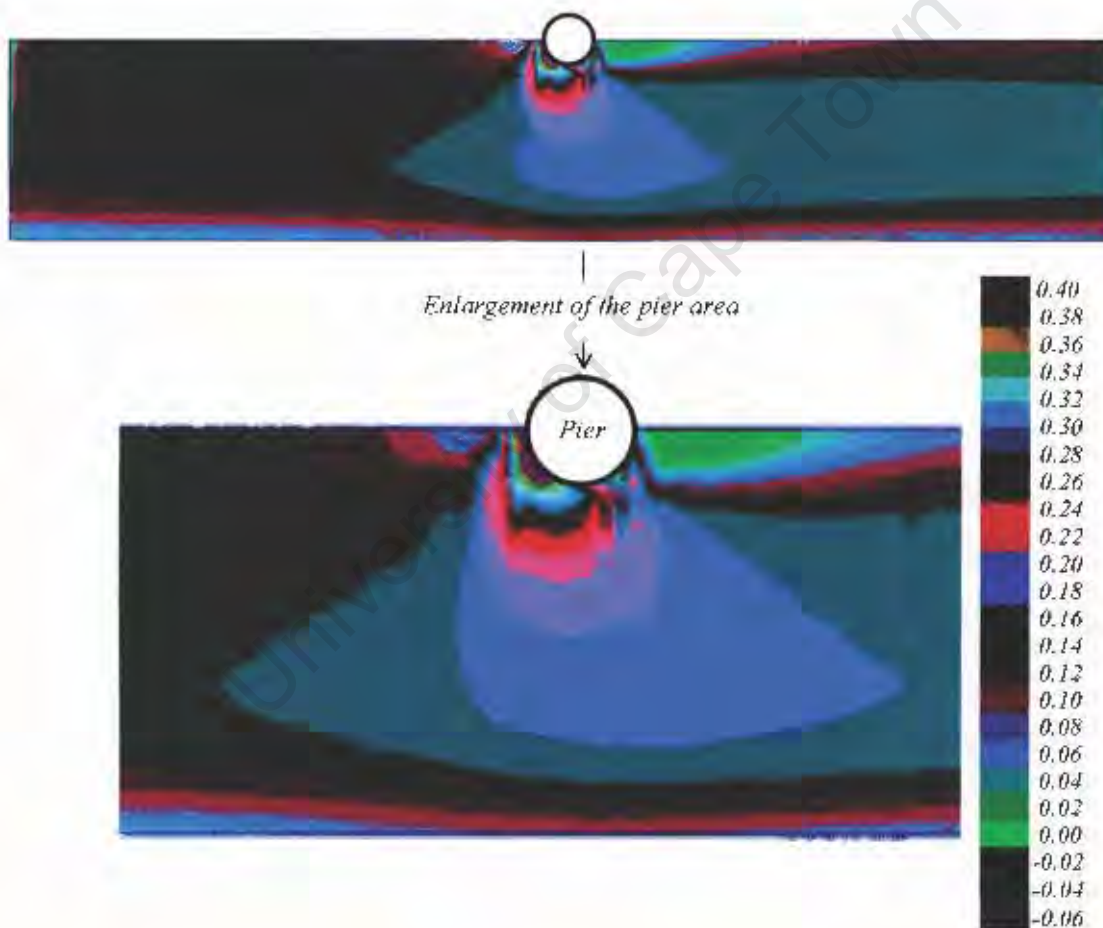


Figure 8.20 : The scour potential distribution around the pier at a depth of 0.5 mm for the initial condition

The final condition model geometry, i.e. after scour has taken place and equilibrium conditions have been reached, is shown in figure 8.21. The uneven channel bed dipped down in the region of the pier to a maximum scour depth of approximately 56 mm.

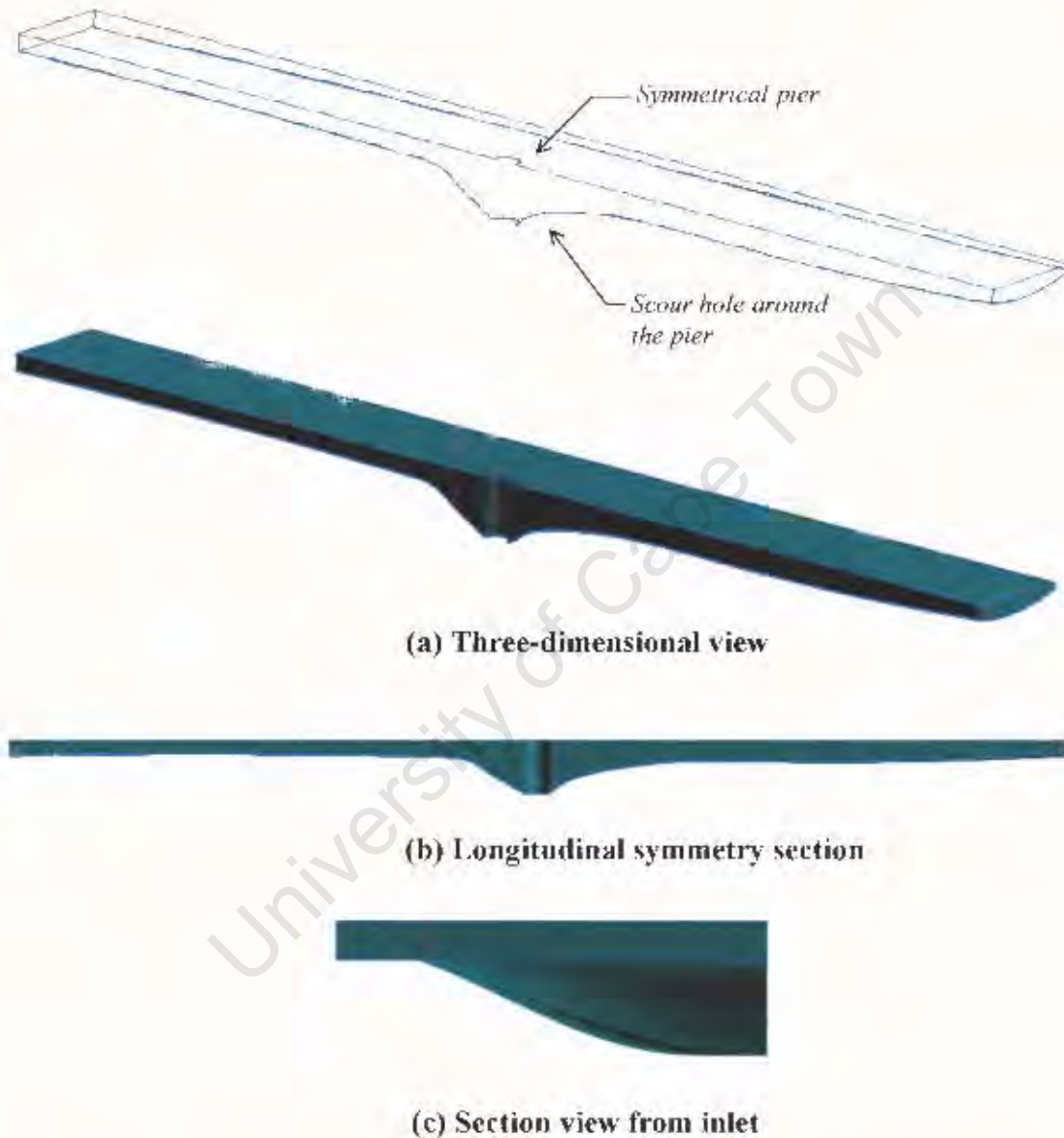


Figure 8.21 : The model geometry for the final equilibrium scour condition

A plan view of the mesh in the vicinity of the pier is shown in figure 8.22. The mesh was refined adjacent to the pier where the scour hole was created.

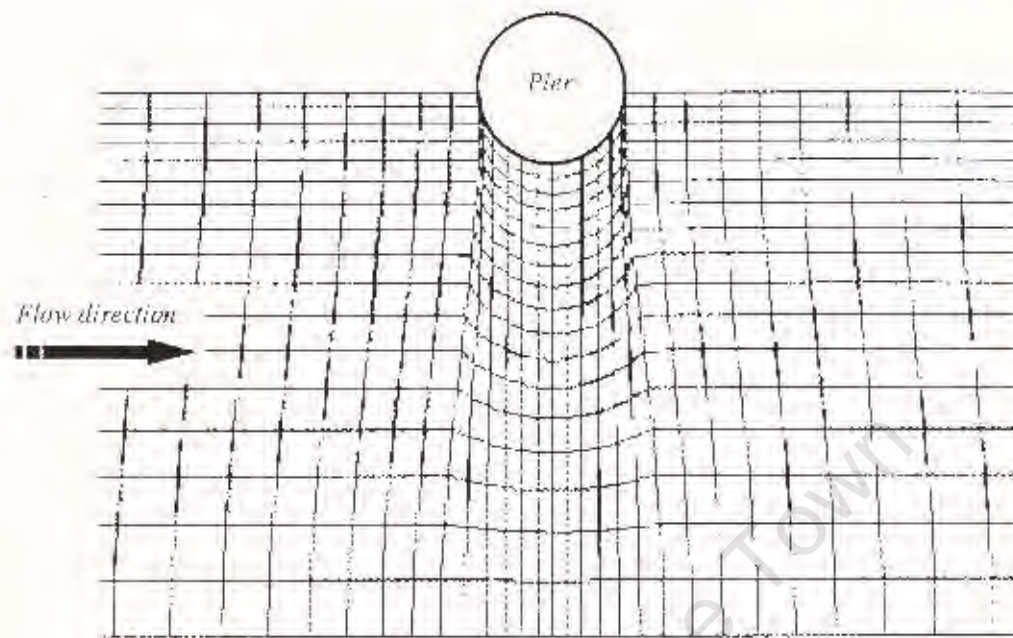


Figure 8.22 : Plan view of the mesh refinement in the vicinity of the pier

The incipient motion critical function (equation 3.48) for sediment on an uneven bed was solved for at a depth of 0.5mm above the channel bed. As the geometry for the final condition was created with a deformed bed, this critical function varies across the channel bed. The Movability Number was also solved for at a depth of 0.5 mm above the channel bed. The critical incipient motion function (equation 3.48) was then subtracted from this to give the scour potential Ω (equation 6.6). The results were plotted in a spatial distribution using IDL (2000).

Figure 8.23 shows the distribution of the Movability Number at the bed. The values are in the range of 0.1 to 0.2, which are less than the range of 0.2 to 0.4 for the initial condition, thus indicating a reduced scour potential.

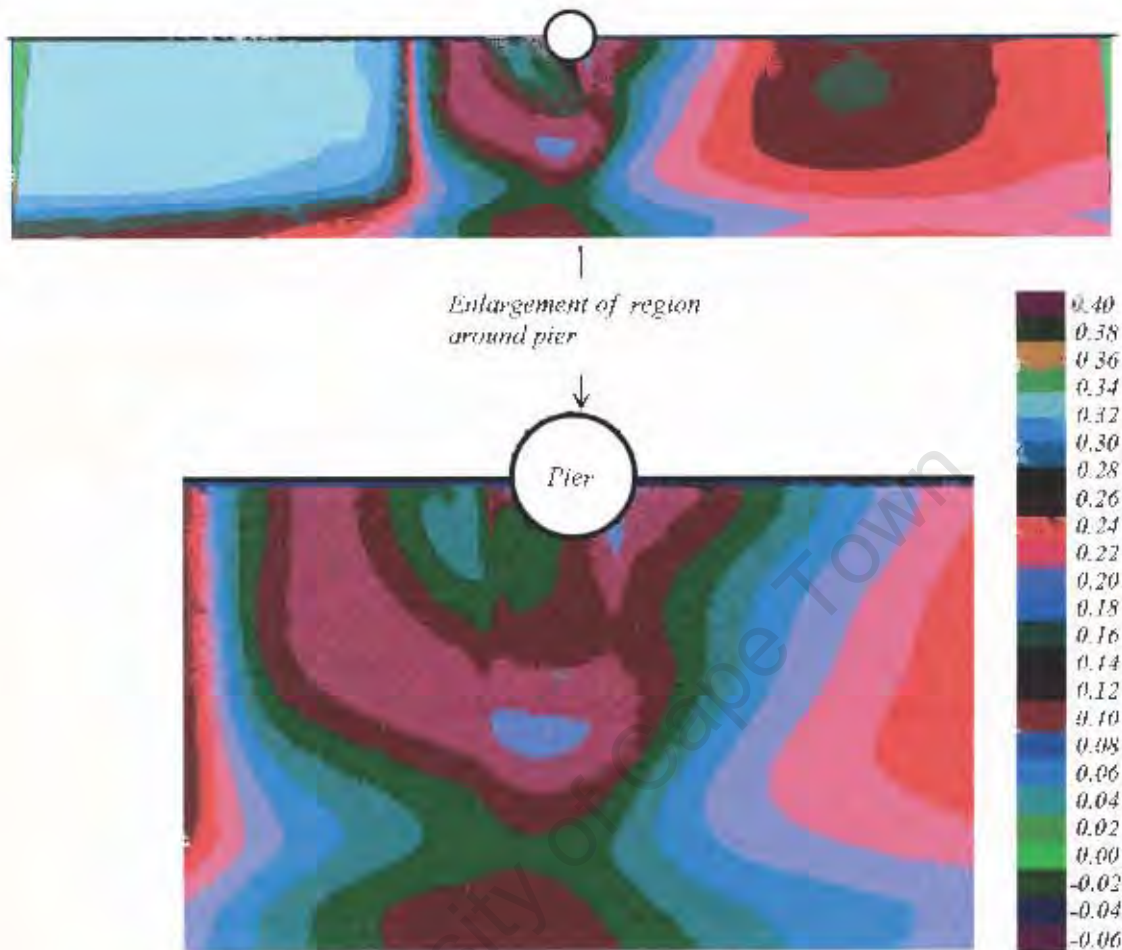


Figure 8.23 : Movability Number distribution for the final condition around the pier

Figure 8.24 shows the distribution of the scour potential (equation 6.6). These values fall in the range -0.04 near the scour hole to 0.06 over the general channel bed. These values are closer to zero than the scour potential for the initial condition i.e. $0.1 - 0.25$, as expected for equilibrium conditions.

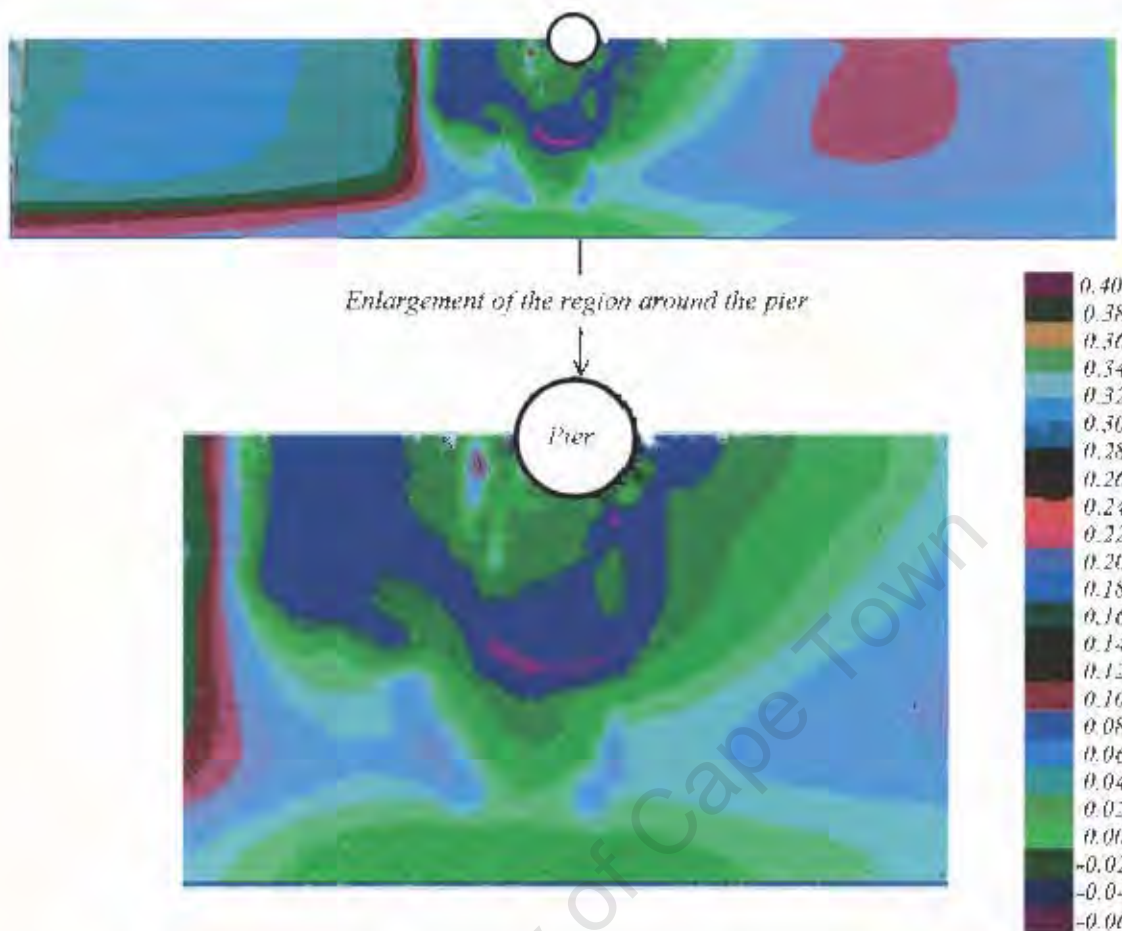


Figure 8.24 : Distribution of the scour potential adjacent to the pier for the final equilibrium profile

8.5 Conclusions

The results of computational scour predictions around various engineering structures have been presented.

The initial and final channel profiles were modeled prior to a vertical weir. The initial conditions illustrated the strong erosive potential of the water, as the applied power values at the channel bed were higher than the incipient motion power requirement. This result was reinforced by the Movability Number distribution, which resulted in a virtual Movability Number greater than 0.17, the critical value for rough boundaries,

throughout a large portion of the depth. For the final equilibrium condition, the applied unit stream power distribution revealed the incipient motion power requirement was just present at the bed. Similarly, the Movability Number only reached the value of 0.17 at the bed.

The scour patterns around a vertical-wall abutment in a level, rectangular channel were predicted with both applied stream power and Movability Number distributions. The predicted scour region compared reasonably well to the observed scour in the laboratory. The scour region predicted by the Movability Number was slightly larger than that predicted by the applied stream power, and therefore a more conservative option.

The scour patterns around a cylindrical pier in a level, rectangular channel were predicted with both applied stream power and Movability Number distributions. The distribution generated by the applied unit stream power predicted the shape of the scour, however the area was shifted towards the line of symmetry. This shift was attributed to the surface being modeled as a horizontal moving wall, which tends to raise the pressure on the line of symmetry above what it would be in reality. In reality a slight drop in surface water elevation and sediment deposition characterises the downstream region. The Movability Number distribution was found to over-predict the scour in the adjacent regions and under-predict the scour where it was observed in the laboratory.

The initial and final condition scour profiles around a cylindrical pier in a self-formed channel with a mobile bed were modeled. These required the introduction of a new function that incorporates the effect of topography. The initial condition revealed large values of the Movability Number i.e. greater than the 0.17 required for incipient motion at a rough boundary. The values of the scour potential (equation 6.6) were in the range 0.1 to 0.25, i.e. larger than zero, indicating a tendency towards scour. In the final condition, a reduction in the Movability Number values occurred at the channel bed. The scour potential (equation 6.6) revealed values in the range of -0.04 to 0.06, with the lowest values occurring in the scour hole region. These low values indicate

that equilibrium conditions are imminent, thus supporting the scour potential, (equation-6.6), as an alternative means of scour prediction.

University of Cape Town

Chapter 9

Conclusions

A numerical model, based on the unit stream power approach to incipient motion, has been developed as an alternative to laboratory experiments for the prediction of scour around engineering structures. This model was tested against laboratory data of observed bed movement and sediment scour profiles. Despite the limitations imposed by the CFD software, the results were sufficiently encouraging to suggest that there is potential for the use of this method for scour prediction given improved software.

9.1 Incipient Motion Criteria

The unit stream power equation for three-dimensional incompressible flow due to gravitational acceleration was employed to predict the incipient motion conditions. This applied stream power was then compared to a particle's independent power requirement (Rooseboom 1974) to determine whether particle movement or suspension would occur for the sand and the beads respectively.

This applied stream power was also expressed in terms of the Movability Number (Rooseboom 1992; Armitage 2001) for the experiments with sand particles. The applied stream power and the Movability Number were employed in the numerical model to generate contours of the predicted bed suspension and the scour patterns around hydraulic structures in rigid, rectangular channels.

The Movability Number was then adapted to incorporate the angle of repose as well as the longitudinal and transverse directions of the flow to the horizontal plane (Armitage 2001). This was used in the numerical model to predict the initial and final scour profiles around a single, cylindrical pier centrally positioned in a self-formed channel.

9.2 Adaptation of CFX-V4.3 to Simulate Open Channel Flow

The commercial CFD flow solver CFX-V4.3 (1999), which solves the three dimensional Navier-Stokes equations using the finite volume method, was adapted to simulate open channel flow conditions and hence give output in a form suitable for scour prediction. It was validated by comparison with benchmark problems such as the lid-driven cavity and published data of flow over dunes. The standard $k-\epsilon$ and the Low Reynolds Number $k-\epsilon$ models were found to provide the best predictions of the turbulent shear stresses for the two- and three-dimensional flow-fields respectively.

Since CFX-V4.3 does not incorporate a free surface algorithm, the free surface was modeled with a moving wall boundary condition, which suppressed the formation of any Vortex Streets, as well as artificially raising some local pressures. The incoming logarithmic velocity distribution was written into the existing software to create a fully developed boundary layer characteristic of uniform open channel flow at the upstream inlet boundary. The turbulent kinetic energy and rate of viscous dissipation distributions were also programmed as part of the inlet boundary condition to ensure the correct upstream shear stress distribution.

It was found that the QUICK scheme provided the most accurate approximation of the velocity in the lid-driven cavity benchmark analysis. The QUICK differencing scheme was therefore implemented for the discretisation of the velocity and pressure terms, while the HYBRID scheme was employed for solution of the turbulent quantities to ensure solutions with positive k or ϵ values.

The steady state simulations gave similar results to the transient solution, and hence the steady state solution was implemented.

CFX-V4.3 is programmed in Fortran 77. Additional Fortran subroutines were written in to the software to solve for the variables in the applied stream power equation, the Movability Number (Liu 1957), the Movability Number adjusted for topography (Armitage 2001) and to incorporate surface roughness at the channel bed. The

resulting numerical model was then used to predict the onset or cessation of particle motion on the channel bed and hence the likelihood of local scour or deposition respectively.

9.3 Predictions of Bead Movement

The movement of beads around engineering structures such as weirs, underflow structures and sudden contractions and expansions was numerically predicted.

The numerical model was found to under-predict the movement of the beads over the three vertical weirs that were simulated. The model did, however, show a correlation to the bead path as observed in the laboratory (Offerman 1998).

Ten underflow structures were numerically simulated. Comparison of the predicted bead movement to the observed movement (Offerman 1998) in the physical experiments revealed similar results. The applied stream power distribution at the water surface attained the bead power requirement for suspension of the largest particles that passed under the structure.

Numerical predictions of bead movements through a sudden contraction and over a weir were simulated for six different configurations. The maximum applied unit stream power at the contraction was less than the bead power requirement for suspension of the largest bead to pass over the weir. The stream power values at the weir, however, were found to correlate well with this power requirement.

The bead movement through a sudden expansion and over a weir yielded similar results to the contractions. The applied stream power contours were under-predicted, while the bead path as observed in the laboratory (Lyons 1999) was well predicted. This was attributed to the stochastic variation in the unit stream power P_i about the mean $\overline{P_i}$ e.g. $P_i = \overline{P_i} + P'_i$ and if P'_i is positive then P_i is sufficient to suspend the bead.

9.4 Prediction of Scour Profiles

The results from the numerical model were tested against scour profiles measured in the laboratory.

The initial and final scour conditions were modeled upstream of a vertical weir. The initial conditions illustrated the strong erosive potential of the water, as the applied power values at the channel bed were higher than the incipient motion power requirement. This result was reinforced by the Movability Number contours, which resulted in virtual Movability Number greater than 0.17, the critical value for rough boundaries, throughout a large portion of the depth. The distribution of the applied unit stream power for the final condition revealed the incipient motion power requirement was just present at the bed. Similarly, the Movability Number only reached the value of 0.17 at the bed.

The scour patterns around a vertical-wall abutment in a level, rectangular channel were numerically predicted with both applied stream power and Movability Number contours. The predicted scour region compared reasonably well to the observed scour in the laboratory.

The scour patterns around a cylindrical pier in a level, rectangular channel were numerically predicted with both applied stream power and Movability Number contours. The contours generated by the applied unit stream power predicted the shape of the scour, however the area was shifted towards the line of symmetry. This shift was attributed to the surface being modeled as a horizontal moving wall, which tends to raise the pressure on the line of symmetry above what it would be in reality. The Movability Number contours were found to over predict the scour in the adjacent regions and under predict the scour where it was observed in the laboratory.

The initial and final condition scour profiles around a cylindrical pier in a self-formed channel with a mobile bed were modelled. The initial condition revealed large values of the Movability Number i.e. greater than the 0.17 required for incipient motion at a rough boundary. The values of the scour potential (equation 6.6) were in the range

0.1 to 0.2, i.e. larger than zero, indicating a tendency towards scour. The final equilibrium condition showed a reduction in the Movability Number values. The scour potential (equation 6.6) revealed values in the range of -0.04 to 0.06, with the lowest values occurring in the scour hole region. These low values indicate that equilibrium conditions are present, thus supporting the scour potential (equation 6.6) as an alternative means of scour prediction.

9.5 Limitations

The CFD software was implemented on a 667MHz Intel Pentium III processor with 256MB of RAM. The computational time, which was on average three hours per analysis, would be reduced with a faster processor. The size of the flow domain and the degree of mesh refinement could be improved with more RAM. An elongated model could allow sufficient length for the boundary layer to develop, and thus eliminate the necessity to program the turbulent kinetic energy, the rate of viscous dissipation and the inlet velocity distributions. These are programmed in to simulate the developed boundary layer in steady, uniform flow.

CFX-V4.3 (1999) does not include an algorithm for free surfaces. Modelling the free surface as a moving wall suppresses the development of vortices around the hydraulic structures, and thus reduces the accuracy of the results. Furthermore, the water level typically rises upstream and drops downstream of structures such as abutments and piers, thus influencing the shear stress distribution. This impacts the power distribution and hence the scour prediction.

Chapter 10

Recommendations for Future Research

Numerical prediction of scour around hydraulic structures in open channel flow provides a possible alternative to time-consuming laboratory experiments. The model presented in this study is sufficiently encouraging to suggest that there is potential for the use of this model for scour prediction given improved software. Future research in the following areas is required:

- (i) In this study the initial and final scour conditions were modelled to reveal the scour potential of the water and the critical incipient motion conditions respectively. This required existing laboratory data to determine the scoured profile. There is hence a need for the gradual development of the scour profile using automated grid generation.
- (ii) CFX-V4.4 (2001) was released in the concluding stages of this research. This package incorporates a free surface algorithm which predicts the surface water profile and open channel characteristics such as vortex development. Implementing this package should improve the shear stress distribution and hence the applied power at the bed and the predicted scour profiles.
- (iii) Increased computing power, i.e. RAM, would enhance the computing efficiency such that limitations on the mesh refinement and the size of the flow domain would be reduced. Increased CPU would increase the solver speed, and hence the number of iterations could be increased to achieve improved convergence within a reasonable time frame.
- (iv) Unit stream power has been shown to be related to sediment concentration (Rooseboom 1992). Improved unit stream power predictions throughout the depth could be used to determine concentration distributions with depth. Integrating this concentration over the depth provides the total sediment load, an essential requirement for design purposes that is currently based on a variety of formulae (details in Chadwick and Morfett 1993).

References

- Ackers P. 1958: Resistance to Fluid Flowing in Channels and Pipes, *Hydraulic Research Paper no. 1, H.M.S.O., London.*
- Ackers P. and White W.R. 1973: Sediment Transport: New Approach and Analysis, *Proc. ASCE*, vol. 99, no. 11, pp 2041-2060.
- Ahmad M. 1953: Experiments on Design and Behaviour of Spur Dikes, *Proc. IAHR Conference*, Minnesota, U.S.A., pp 145-159.
- Annandale G.W. 1986: Reservoir Sedimentation: the Stable Non-equilibrium State, Technical Paper, *The Civil Engineer in South Africa.*
- Armitage N.P. 1998: The Removal of Urban Litter from Stormwater Conduits and Streams, *WRC Report no. TT 95/98.*
- Armitage N.P. 2001: *A Unit Stream Power Theory for the Prediction of Scour and Deposition*, PhD Thesis, The University of Cape Town (in preparation).
- Babaeyan-Koopaei K. 1996: *A Study of Straight Channels and their Interaction with Bridge Structures*, Ph.D. Thesis, Newcastle.
- Bagnold R.A. 1960: Sediment Discharge and Stream Power, *U.S. Geological Survey Circular 421*, Washington D.C. pp23.
- Bagnold R.A. 1966: An Approach to the Sediment Transport Problem for General Physics, *Geological Survey Professional Paper 422-1*, Washington D.C.
- Baojun S., Chuangang G. and Zhengxian L. 1996: On Two-equation Turbulence Model for Complex Turbulent Flows by using the D.I. Approximation: I Theory in 'Flow Modelling and Turbulent Measurements *', pp 315-322.
- Biglari B. and Sturm T.W. 1998: Numerical Modelling of Flow Around Bridge Abutments in Compound Channels, *ASCE, Journal of Hydraulic Engineering*, vol. 124, no. 2, pp156-164.
- Boussinesq J.V. 1877: Essai sur la 'Theorie des Eaux Courantes (Essay on the theory of waterflow), *Memoires Presentes par Diverss savants a l'academic des Sciences*, vol.3, no.1, Paris, France.
- Breusers H.N.C. 1966: Conformity and Time-scale in Two-dimensional Local Scour, *Proc. Symposium on Model and Prototype Conformity:1-8*, Hydraulic Research Laboratory, Poona (also Delft Hydraulics, Delft, Publication 40).
- Breusers H.N.C., Nicolet G. and Shen H.W. 1977: Local Scour Around Cylindrical Piers, *Journal of Hydraulic Research, IAHR*, vol. 15, no. 3, pp 211-252.
- Breusers H.N.C. and Raudkivi A.J. 1991: *Scouring*, A.A. Balkema, Rotterdam, Brookfield.

- Brooks N.H. 1955: Mechanics of Streams with Movable Beds of Fine Sands, *Proc. of the ASCE*, vol. 81, no. 668, pp 1-208.
- Cellino M. and Graf W.H. 2000: Experiments on Suspension Flow in Open Channels with Bed Forms, *Journal of Hydraulic Engineering Research*, vol. 38, no. 4.
- CFX 1999: CFX Version 4.3, *AEA Technology*.
- CFX 2001: CFX Version 4.4, *AEA Technology*.
- CFX Manual 1999: CFX Version 4.3, *AEA Technology*.
- Chabard J.P., Archambeau F., Laurence D. and Pot G. 1996: Recent Advances in Modeling Turbulent Flows using a Finite Element Method in 'Flow Modelling and Turbulent Measurements *', pp 3-16.
- Chadwick A. and Morfett J. 1993: *Hydraulics in Civil and Environmental Engineering*, second edition, E&FN Spon, Great Britain.
- Chanson H. 1999: *The Hydraulics of Open Channel Flow – An Introduction*, J.W. Arrowsmith Ltd., Bristol, Great Britain.
- * Chen C.J., Shih C., Lienau J. and Kung R.J. 1996: *Flow Modelling and Turbulence Measurements*, A.A. Balkema, Rotterdam, Brookfield.
- Chen H-C. and Wang L. 1996: Second Moment RANS Calculations of Turbulent Flows in Curved Channels in 'Flow Modelling and Turbulent Measurements *', pp 425-432.
- Cheng N-S. 1997: Effect of Concentration on Settling Velocity of Sediment Particles, *ASCE, Journal of Hydraulic Engineering*, vol. 123, no. 8, pp 728-731.
- Cheng N-S. 1997: A Simplified Settling Velocity Formula for Sediment Particles, *ASCE, Journal of Hydraulic Engineering*, vol. 123, pp 149-152.
- Cheng N-S. and Lim S-Y. 1998: Prediction of Live-bed Scour at Bridge Abutments, *ASCE, Journal of Hydraulic Engineering*, vol. 124, no. 6.
- Chien N. and Wan Z. 1998: Mechanics of Sediment Transport, *American Society of Civil Engineers Press*, Virginia.
- Cokljat D. and Kralj C. 1997: On Choice of Turbulence Model for Prediction of Flows over River Bed Forms, *Journal of Hydraulic Research*, vol. 35, no. 3.
- Coles D. 1956: The Law of the Wake in the Turbulent Boundary Layer, *Journal of Fluid Mechanics*, vol.1, pp 191-226.
- Compion J.K. 1997: *The Trapping of Litter in Waterways by means of Flow Retardation and Screens*, The University of Stellenbosch.

- Corey A.T. 1949: *Influence of Shape on the Fall Velocity of Sand Grains*, MSc Thesis, Colorado Agricultural and Mechanical College, Fort Collins, Colorado, U.S.A.
- Dey S., Bose K. and Sastry G.L.N. 1995: Clear-water Scour at Circular Piers: A Model, *ASCE, Journal of Hydraulic Engineering*, vol. 121, no. 12.
- Dietz J.W. 1969: *Kolkbildung in Fein- oder Leichten Sohlmaterialien bei Stromendem Abfluß*, Mitteilungen, Heft 155, Universitat Fridericiana Karlsruhe.
- Durst F. and Rastogi A.K. 1979: Theoretic and Experimental Investigations of Turbulent Flows with Separation, *Turbulent Shear Flows I*, Springer Verlag, Heidelberg.
- Einstein H.A. 1950: The Bed Load Function for Sediment Transportation in Open Channel Flow, *Technical bulletin no. 1026*, U.S. Department of Agriculture, Washington D.C.
- Einstein H.A. and Chien N. 1954: Second Approximation to the Solution of the Sediment Suspended Load Theory, University of California and Missouri river Division, U.S. Army Corps of engineers, *MRD Sediment Service*, no. 2.
- Ettema R.E. 1980: Scour at Bridge Abutments, *Report no. 54*, School of Engineering, The University of Auckland, New Zealand.
- Ettema R.E., Mostafa E.A., Melville B.W. and Yalin A.A. 1998: Local Scour at Skewed Piers, *Journal of Hydraulic Engineering*, vol. 124, no. 7, pp 756-759.
- Ferziger J.H. and Milouan P. 1996: *Computational Methods for Fluid Dynamics*, Springer-Verlag, Berlin, Heidelberg.
- Feuilloley D. 1999: *Transport of Sediment through Contractions in Rapidly Varied Flow*, The University of Cape Town.
- Fiorotto V. 1992: 'Local Scour at Bridge Abutments' B.W. Melville- Discussion, *ASCE, Journal of Hydraulics*, vol.118, no.4.
- Fletcher D.F. 2000: Personal Correspondence with Dr. David F. Fletcher, CFX Technical Support Manager for Australasia and S.E. Asia, ATD International, and Honoray Academic, Department of Chemical Engineering, University of Sydney, Australia.
- Fortier S. and Scobey F.C. 1926: Permissible Canal Velocities, *Transactions of the ASCE*, vol. 89.
- Fujisaki K., Tanaka K. and Shiono K. 1999: Concentration Profiles of Suspended Sediment in Rectangular Open Channel Flow, *IAHR Symposium on River, Coastal and Estuarine Morphodynamics*, Geneva.

- Ghetti A. 1954: *The Study of Bed Erosion at Weirs by means of Small-scale Models*, - (in Italian), University of Padua, Institute of Hydraulics, Paper no. 167.
- Ghia U., Ghia K.N. and Shin C.T. 1982: High-Re Solutions for Incompressible Flow using the Navier-Stokes Equations and a Multi-grid, *Journal of Computational Physics*, vol. 48, pp 387-411.
- Gosman D.A. and Pun W.M. 1973: Lecture Notes for Course Entitled 'Calculation of Recirculating Flow', *Report HTS/74/2*, Heat Transfer Section, Imperial College, London, England.
- Hancu S. 1971: Sur le Calcul des Affouillements locaux Dams la Zone des Piles de Ponts, *Proc. 14th IAHR Congress*, vol. 3, pp 299-313.
- Harlow F.H. and Welch J.E. 1965: Numerical Calculation of Time-dependent Viscous Incompressible Flow of Fluid with Free Surface, *Physics of Fluids*, vol. 8, pp 2182-2189.
- Harnett B.R. 1998: *Damming of the River Sabie, South Africa: The Sedimentation Problem*, Internal Report, Centre for Water and the Environment, University of the Witwatersrand.
- Heywood H. 1938: Measurement of the Fineness of Powdered Materials, *Proc. Institute of Mechanical Engineering*, vol. 140.
- Hinze J.D. 1962: Turbulent Pipe Flow, *Mecanique de la Turbulence*, 108, Editions du Centre National de la Recherche Scientifique, Paris, France, pp 129-165.
- Hjulstrom F. 1935: The Morphological Activity of Rivers as Illustrated by River Fyris, *Bulletin of the Geological Institute*, Uppsala, vol. 25, chapter 3.
- Hoang M.L., Verboven P., De Baerdemaeker J. and Nicolai B.M. 1999: Analysis of the Air Flow in a Cold Store by means of Computational Fluid Dynamics, *International Journal of Refrigeration*, vol. 23, pp 127-140.
- Hoffmans G.J.C.M. and Verheij H.J. 1997: *Scour Manual*, A.A. Balkema, Rotterdam.
- Hopkins G.R., Vance R.W. and Kasraie B. 1980: Scour Around Bridge Piers, *Report FHWA-RD-79-103*, Federal Highway Administration, Offices of Research and Development Environmental Division, Washington D.C.
- Hsich W.D. and Chang K.C. 1996: Some Numerical Considerations in Turbulent Flow Calculation incorporated with Reynolds Stress Models in 'Flow Modelling and Turbulent Measurements *', pp 441-447.
- IDL 2000: Interactive Data Language Version 5.2, product of *Research Systems*, Inc. (RSI).

- Jain S.C. and Fischer E.E. 1979: Scour around Circular Bridge Piers at High Froude Numbers, *Report no. FHWA-RD-79-104*, Federal Highway Administration, Washington D.C.
- Jaw S-Y. and Chen C-J. 1996: Present Status of k- ϵ Models and their Achievements in 'Flow Modelling and Turbulent Measurements *' pp 295-304.
- Johnson P.A. 1995: Comparison of Pier-scour Equations using Field Data, *ASCE, Journal of Hydraulic Engineering*, vol. 121, no. 8.
- Kandasamy J.K. 1989: Abutment Scour, *Report no. 458*, School of Engineering, The University of Auckland, New Zealand.
- Karim O.A. and Ali K.H.M. 2000: Prediction of Flow in Local Scour Holes caused by Turbulent Water Jets, *Journal of Hydraulic Research*, vol. 38, no. 4.
- Kazanskij I. 1981: *Über Theoretische und praxis bezogene Aspekte des Hydraulischen Feststofftransportes*, University of Hanover, Mitt. Des Franzius-Inst., vol. 52.
- Kovacs A.E. 1998: Prandtl's Mixing Length Concept Modified for Equilibrium Sediment-laden Flows, *ASCE, Journal of Hydraulic Engineering*, vol. 128, no. 8.
- Kuhnle R.A., Alonso C.V. and Shields F.D. 1999: Geometry of Scour Holes Associated with 90 degree Spur Dikes, *ASCE, Journal of Hydraulic Engineering*, vol. 125, no. 9.
- Laursen E.M. 1963: An Analysis of Relief Bridge Scour, *ASCE, Journal of the Hydraulics Division*, vol. 89, no. 3.
- Li W-H. 1955: Criteria for Similitude of Scour Hole below Hydraulic Structures, *Proc. 6th IAHR Congress*, The Hague, Paper C4.
- Lim S-Y. 1997: Equilibrium Clear-water Scour around an Abutment, *ASCE, Journal of Hydraulic Engineering*, vol. 123, no. 3.
- Liu H.K. 1957: Mechanics of Sediment – Ripple Formation, *Journal of the Hydraulics Division, ASCE*, vol. 83, no. 2, pp 1197(1-23).
- Liu H.K. 1958: Closure to 'Mechanics of Sediment – Ripple Formation', *Journal of the Hydraulics Division, ASCE*, vol. 84, no. 5, pp 10-12.
- Long C.E., Wiberg P.L. and Nowell A.R.M. 1993: Evaluation of von Karman's Constant from Integral Flow Parameters, *ASCE, Journal of Hydraulic Engineering*, vol. 119, no. 10.
- Lyons S. 1999: *Sediment Entrainment and Deposition at Sudden Expansions in Open Channel Flow*, BSc Thesis, The University of Cape Town.
- Manning R. 1980: On the Flow of Water in Open channels and Pipes, *Institution of Civil Engineers*, Ireland.

- Mavis F.T. and Laushey L.M. 1948: A Reappraisal of the Beginning of Bed-movement Competent Velocity, *International Association for Hydraulic Research*, second meeting, Stockholm.
- Melville B.W. 1974: Scour at Bridge Sites, University of Aucland, New Zealand, School of Engineering, *Report no 104 or Proc. NZ National Roads Board*, PRU Summer Meeting on Bridge Design and Research.
- Melville B.W. 1975: *Local Scour at Bridge Sites*, University of Aucland, New Zealand, Ph.D. Thesis, Dept. of Civil Engineering, Rep. No. 117.
- Melville B.W. and Raudkivi A.J. 1977: Flow Characteristics in Local Scour at Bridge Piers, *Journal of Hydraulic Engineering*, vol. 15, no. 4, pp 373-380.
- Melville B.W. and Sutherland A.J. 1988: Design Method for Local Scour at Bridge Sites, *ASCE, Journal of Hydraulic Engineering*, vol. 114, no. 10, pp 1210-1226.
- Melville B.W. 1992: Local Scour at Bridge Abutments, *Journal of Hydraulic Engineering*, vol. 118, no. 4, 615-631.
- Melville B.W. 1997: Pier and Abutment Scour: Integrated Approach, *ASCE, Journal of Hydraulic Engineering*, vol. 123, no. 2, pp 125-136.
- Melville B.W. and Chiew Y.M. 1999: Time Scale for Local Scour at Bridges, *ASCE, Journal of Hydraulic Engineering*, vol. 125, no. 1, pp 59-65.
- Meyer-Peter P.E. and Muller R. 1948: Formulas for Bed Load Transport, *Proc. of the 3rd International Association for Hydraulic Research*, Stockholm, pp 39-64.
- Mitchell A. 2000: *Sediment Transport around Abutments in Open Channel Flow*, BSc Thesis, The University of Cape Town.
- Midgely M. 2000: *Sediment Movement Around Piers in Open Channel Flow*, BSc Thesis, University of Cape Town, South Africa.
- Molls T. and Chaudhry M.H. 1995: Depth-averaged Open-channel Flow Model, *ASCE, Journal of Hydraulic Engineering*, vol. 121, no. 6, pp 453-465.
- Mosonyi E. and Schoppman B. 1968: *Ein Beitrag Erforschung von ortlichen Auskolkungen hinter geneigten Befestigungsstrecken in Abhangigkeit*, Mitteilungen des Theodor-Rehbock-FluBbaulaboratoriums, Universitat Karlsruhe.
- Neill C.R. 1973: *Guide to Bridge Hydraulics*, University of Toronto Press.
- Nezu I. And Nakagawa H. 1993: *Turbulence in Open Channel Flows*, A.A. Balkema, Rotterdam, Netherlands.

- Nikuradse J. 1933: Stromungsgesetze in Rauben Rohren (Laws of turbulent pipe flow in rough pipes), *VDI, Forschungsheft, no. 361*, (in German – translated in NACA Tech. Memo no. 1292, 1950)
- Offerman J. 1998: *Sediment Transport in Two-dimensional Flow with Fixed Boundaries*, BSc Thesis, The University of Cape Town.
- Olsen N.R.B. and Melaaen M.C. 1993: Three-dimensional Calculation of Scour Around Cylinders, *Journal of Hydraulic Engineering*, vol. 119, no. 9, pp 1044-1054.
- Ouillon S. and Dartus D. 1997: Three-dimensional Computation of Flow around a Groyne, *ASCE, Journal of Hydraulic Engineering*, vol. 123, no. 11, pp 962-970.
- Panchang V.G. and Richardson J.E. 1998: Three-dimensional Simulation of Scour-inducing Flow at Bridge Piers, *ASCE, Journal of Hydraulic Engineering*, vol. 124, no. 5, pp 530-540.
- Perzon s., Davidson L. and Ramnefors M. 1996: Reynolds Stress Modeling of Complex Flows over Curved Surfaces in 'Flow Modelling and Turbulent Measurements *', pp 407-414.
- Prandtl L. 1925: Uber die Ausgebildete Turbulenz (On fully developed turbulence), *Z.A.M.M.*, vol. 5, pp 136-139, (in German).
- Rastogi A.K. and Rodi W. 1978: Calculation of General Three-dimensional Boundary Layers, *AIAA Journal*, vol. 16, pp 151-159.
- Raudkivi A.J. and Ettema R. 1983: Clear-water Scour at Cylindrical Piers, *ASCE, Journal of Hydraulic Engineering*, vol. 109, no. 3, pp 338-349.
- Raudkivi A.J. 1986: Functional Trends of Scour at Bridge Piers, *ASCE, Journal of Hydraulic Engineering*, vol. 112, no. 1, pp 1-13.
- Raudkivi A.J. 1998: *Loose Boundary Hydraulics*, A.A. Balkema, Rotterdam, Netherlands.
- Reynolds A.J. 1974: *Turbulent Flows in Engineering*, Wiley and Sons.
- Richardson E.V. and Richardson J.R. 1992: 'Local Scour at Bridge Abutments' B.W. Melville- Discussion, *ASCE, Journal of Hydraulics*, vol.118, no.4.
- Richardson E.V., Richardson J.R., Davis S.R. and Harrison L.J. 1993: Evaluating Scour at Bridges, *HI-90-017, HEC-18, Federal Highway Administration*, McLean, Va.
- Richardson E.V. and Richardson J.R. 1993b: Discussion on 'Local Scour at Bridge Abutments' by B.W. Melville, *ASCE, Journal of Hydraulic Engineering*, vol. 119, no. 9, 1069-1071.

- Roberson J.A. and Crowe C.T. 1997: *Engineering Fluid Mechanics*, sixth edition, - John Wiley and Sons, Inc., USA and Canada.
- Rodi W. 1984: *Turbulence Models and their Application in Hydraulics – A State of the Art Review*, IAHR, Delft, Netherlands.
- Rodi R.W. and Celik I. 1984: Simulation of Free-surface Effects on Turbulent Channel Flows, *Physico-chem. Hydrodynam.*, vol. 4, pp 544-560.
- Rooseboom A. 1974: Open Channel Fluid Mechanics, *Report no. 62, Department of Water Affairs*, Pretoria, South Africa.
- Rooseboom A. 1975: *Sediment Transport in Rivers and Reservoirs*, DEng dissertation, University of Pretoria, South Africa (in Afrikaans).
- Rooseboom A. and Le Grange P. 1992: The Development of a Model to Simulate Channel Deformation in Alluvial Channels, *Report to Water Research Commission of South Africa*.
- Rooseboom A. 1992: Sediment Transport in Rivers and Reservoirs – A Southern African Perspective, *Report to Water Research Commission, No. 297/1/92*, Sigma Beta Consulting Engineers.
- Rooseboom A. 2001: Personal Communication with Professor A. Rooseboom, University of Stellenbosch, Western Cape.
- Rouse H. 1939: An Analysis of Sediment Transportation in the Light of Fluid Turbulence, *Soil Conservation Service Report no. SCS-TP-25*, U.S. Department of Agriculture, Washington D.C.
- Rubey W.W. 1933: Settling Velocities of Gravel, Sand and Silt Particles, *American Journal of Science*.
- Schiller L. and Naumann A. 1933: Über die Grundlegenden Berechnungen bei der Schwerkrafttaubereitung, *Zeitschrift der VDI*, vol. 77.
- Schlichting H. 1979: *Boundary Layer Theory*, seventh edition, Mc Graw-Hill Inc., United States of America.
- Shen H.W., Scheider V.R. and Karaki S. 1969: Local Scour around Bridge Piers, *Journal of the Hydraulics Division*, vol. 95, no. 6, pp 1919-1940.
- Shields A. 1936: *Application of Similarity Principles and Turbulence Research to Bed Load Movement*, California Institute of Technology, Pasadena (translated from German).
- Simoes F.J.M. 1996: An Algebraic Subgrid-scale RNG Turbulence Model for Smooth Free-surface Flows in '*Flow Modelling and Turbulent Measurements **', pp 687-694.

- Simons D.B. and Richardson E.V. 1966: Resistance to Flows in Alluvial Channels, - *U.S. Geological Survey Professional Paper 422-J*.
- Simons D.B. and Senturk F. 1977: *Sediment Transport Technology*, USA Water Resource Publications, Fort Collins, Colorado.
- Singhal A.K. and Spalding D.B. 1975: Prediction of Two-dimensional Boundary Layer with the Aid of the k- ϵ Model, *Mechanical Engineering Report HTS/75*, Imperial College.
- Solitiropoulos F. and Ventikos Y. 1996: Assessment of some Non-linear Two-equation Turbulence Models in a 3D Shear Flow in '*Flow Modelling and Turbulent Measurements **', pp 331-338.
- Speziale C.G. 1991: Analytical Methods for the Development of Reynolds-stress Closures in Turbulence, *Annual Revision Fluid Mechanics*, vol. 23, pp 107-157.
- Stephenson P.L. 1976: Theoretical Study of Heat Transfer in Two-dimensional Turbulent Flow in a Circular Pipe and between Parallel and Diverging Plates, *International Journal of Heat and Mass Transfer*, vol. 19, pp 413-423.
- Stokes G. 1851: On the Effects of Internal Friction of Fluids on the Motion of Pendulums, *Transactions of the Cambridge Philosophical Society*, vol. 9, part 2, pp 8-106.
- Sturm T.W. and Janua V.S. 1994: Clear-water Scour Around Abutments in Floodplains, *ASCE, Journal of Hydraulic Engineering*, vol. 120, no. 8, pp 956-972.
- Takaki S., Tu h. and Ichiyama M. 1996: 3D Simulations of Flows around Bridge Piers at a River Bend in '*Flow Modelling and Turbulent Measurements **', pp 565-572.
- Tingsanchali T. and Maheswaran S. 1990: 2-D Depth-averaged Flow Computation near Groyne, *ASCE, Journal of the Hydraulics Division*, vol. 116, no. 1, pp 71-86.
- Tominaga A., Liu J. and Nagao M. 1996: Numerical Study on Turbulent Structure over Square and Circular Strip Roughness in Open Channel Flows in '*Flow Modelling and Turbulent Measurements **' pp 505-512.
- Van der Meulen T. and Vinje J.J. 1975: Three-dimensional Local Scour in Non-cohesive Sediments, *Proc. 16th IAHR Congress*, Sao Paulo, Brasil (also Delft Hydraulics, Delft, Publication 180).
- Vanoni V.A. 1975-7(reprint): Sedimentation Engineering, *ASCE Task Committee for the Preparation of the Manual of Sedimentation Committee of the Hydraulics Division*.
- Van Rijn L.C. 1993: *Principles of Sediment Transport in Rivers, Estuaries and Coastal Seas*, Aqua Publications, Netherlands.

- Versteeg H.K. and Malalasekera W. 1995: *An Introduction to Computational Fluid Dynamics*, Longman, Malaysia.
- Wallin S. 1996: A New Explicit Algebraic Reynolds Stress Turbulence Model including an Improved Near-wall Treatment in '*Flow Modelling and Turbulent Measurements **', pp 399-406.
- White C.M. 1940: The Equilibrium of Grains on the Bed of an Alluvial Channel, *Proc. of the Royal Society of London, Series A*, vol. 174, pp 332-338.
- White F.M. 1991: *Viscous Fluid Flow*, second edition, Mc Graw-Hill, Inc.
- White W.R., Paris E. and Bettess R. 1981: Tables for the Design of Stable Alluvial Channels, *HRS, Report no. IT 208*, Wallingford, England.
- Wilcox D.C. 1993: *Turbulence Modeling for CFD*, DCW Industries, LaCanada, CA.
- Yabosaki S.B. and Ponce V.M. 1981: Modelling Circulation in Depth-averaged Flow, *ASCE, Journal of the Hydraulic Division*, vol. 107, no. 11, pp 1501-1518.
- Yakhot V. and Orszag S. 1996: Renormalization Group Analysis of Turbulence I Basic Theory, *Journal Sci. Comput.*, vol. 1, no. 1, pp 3-51.
- Yalin M.S. 1972: *Mechanics of Sediment Transport*, Pergamon Press.
- Yang C.T. 1972: Unit Stream Power and Sediment Transport, *Journal of the Hydraulics Division, Proc. ASCE*, vol. 98, no.10, pp 1805-1820.
- Yang C.T. 1976: Minimum Stream Power and Fluvial Hydraulics, *Journal of the Hydraulics Division, Proc. ASCE*, vol. 102, no.7, pp919.
- Yang C.T. and Song C.S. 1979: Theory of Minimum Rate of Energy Dissipation, *Journal of the Hydraulic Division*, no. 7, pp 769-784.
- Yang C.T. 1979: Unit Stream Power Equations for Total Load, *Journal of Hydrology*, vol.40, 123-138.
- Yang C.T. and Molinas A. 1982: Sediment Transport and the Unit Stream Power Function, *Journal of the Hydraulics Division, Proc. ASCE*, vol. 108, no.6, pp 776-793.
- Yang C.T. 1996: *Sediment Transport – Theory and Practice*, Mc-Graw Hill Inc., Singapore.
- Yanmaz A.M. and Altinbilek H.T. 1991: Study of Time-dependent Local Scour around Bridge Piers, *ASCE, Journal of Hydraulic Engineering*, vol. 117, no. 10, pp 1247-1268.

Zaghloul N. and Mc Corquodale J.A. 1973: A Numerical Model for Flow Past a Spur Dike, *Proc. First Canadian Hydraulic Conference*, Canadian Society for Civil Engineering.

Zanke U. 1982: *Grundlagen der Sedimentbewegung*, Springer – Verlag.

Zegzhda A.P. 1938: *Theoriia Podobija Metodika Rascheta Gidrotekhnicheskikh Modela* (Theory of similarity and methods of design for hydraulic engineering), Gosstroizdat, Leningrad.

University of Cape Town

Appendix

A1 Subroutine USRTRN for velocity gradients between parallel plates

```

SUBROUTINE USRTRN(U,V,W,P,VFRAC,DEN,VIS,TE,ED,RS,T,H,RF,SCAL,
+             XP,YP,ZP,VOL,AREA,VPOR,ARPOR,WFACT,CONV,IPT,
+             IBLK,IPVERT,IPNODN,IPFACN,IPNODF,IPNODB,IPFACB,
+             WORK,IWORK,CWORK)
C
C*****
C
C  USER SUBROUTINE TO ALLOW USERS TO MODIFY OR MONITOR THE SOLUTION AT
C  THE END OF EACH TIME STEP
C  THIS SUBROUTINE IS CALLED BEFORE THE START OF THE RUN AS WELL AS AT
C  THE END OF EACH TIME STEP
C*****
C
C  THIS SUBROUTINE IS CALLED BY THE FOLLOWING SUBROUTINES
C  CUSR TRNMOD
C
C*****
C  CREATED
C  27/04/90 ADB
C  MODIFIED
C  05/08/91 IRH NEW STRUCTURE
C  01/10/91 DSC REDUCE COMMENT LINE GOING OVER COLUMN 72.
C  29/11/91 PHA UPDATE CALLED BY COMMENT, ADD RF ARGUMENT,
C             CHANGE LAST DIMENSION OF RS TO 6 AND IVERS TO 2
C  05/06/92 PHA ADD PRECISION FLAG AND CHANGE IVERS TO 3
C  03/07/92 DSC CORRECT COMMON MLTGRD.
C  23/11/93 CSH EXPLICITLY DIMENSION IPVERT ETC.
C  03/02/94 PHA CHANGE FLOW3D TO CFDS-FLOW3D
C  22/08/94 NSW MOVE TF(IUSED,EQ.0) RETURN' OUT OF USER AREA
C  19/12/94 NSW CHANGE FOR CFX-F3D
C  02/07/97 NSW UPDATE FOR CFX-4
C  02/07/99 NSW INCLUDE NEW EXAMPLE FOR CALCULATING FLUX OF A
C             SCALAR AT A PRESSURE BOUNDARY
C
C*****
C
C  SUBROUTINE ARGUMENTS
C
C  U   - U COMPONENT OF VELOCITY
C  V   - V COMPONENT OF VELOCITY
C  W   - W COMPONENT OF VELOCITY
C  P   - PRESSURE
C  VFRAC - VOLUME FRACTION
C  DEN  - DENSITY OF FLUID
C  VIS  - VISCOSITY OF FLUID
C  TE   - TURBULENT KINETIC ENERGY
C  ED   - EPSILON
C  RS   - REYNOLD STRESSES
C  T    - TEMPERATURE
C  H    - ENTHALPY
C  RF   - REYNOLD FLUXES
C  SCAL - SCALARS (THE FIRST 'NCONC' OF THESE ARE MASS FRACTIONS)

```

```

C  XP  - X COORDINATES OF CELL CENTRES
C  YP  - Y COORDINATES OF CELL CENTRES
C  ZP  - Z COORDINATES OF CELL CENTRES
C  VOL  - VOLUME OF CELLS
C  AREA - AREA OF CELLS
C  VPOR - POROUS VOLUME
C  ARPOR - POROUS AREA
C  WFACT - WEIGHT FACTORS
C  CONV - CONVECTION COEFFICIENTS
C
C  IPT  - 1D POINTER ARRAY
C  IBLK - BLOCK SIZE INFORMATION
C  IPVERT - POINTER FROM CELL CENTERS TO 8 NEIGHBOURING VERTICES
C  IPNODN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING CELLS
C  IPFACN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING FACES
C  IPNODF - POINTER FROM CELL FACES TO 2 NEIGHBOURING CELL CENTERS
C  IPNODB - POINTER FROM BOUNDARY CENTERS TO CELL CENTERS
C  IPFACB - POINTER FROM BOUNDARY CENTERS TO BOUNDARY FACES
C
C  WORK - REAL WORKSPACE ARRAY
C  IWORK - INTEGER WORKSPACE ARRAY
C  CWORK - CHARACTER WORKSPACE ARRAY
C
C  SUBROUTINE ARGUMENTS PRECEDED WITH A '*' ARE ARGUMENTS THAT MUST
C  BE SET BY THE USER IN THIS ROUTINE.
C
C  NOTE THAT OTHER DATA MAY BE OBTAINED FROM CFX-4 USING THE
C  ROUTINE GETADD, FOR FURTHER DETAILS SEE THE VERSION 4
C  USER MANUAL.
C
C *****
C
C  LOGICAL LDEN,LVIS,LTURB,LTEMP,LBUOY,LSCAL,LCOMP
C  + ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
C  CHARACTER*(*) CWORK
C
C+++++ USER AREA 1 ++++++
C
C---- AREA FOR USERS EXPLICITLY DECLARED VARIABLES
C
C+++++ END OF USER AREA 1 ++++++
C
COMMON
+ /ALL/ NBLOCK,NCELL,NBDRY,NNODE,NFACE,NVERT,NDIM
+ /ALLWRK/ NRWS,NIWS,NCWS,IWRFRE,IWIFRE,IWCFRE
+ /ADDIMS/ NPHASE,NSCAL,NVAR,NPROP
+ ,NDVAR,NDPROP,NDXNN,NDGEOM,NDCOEF,NILIST,NRLIST,NTOPOL
+ /CHKUSR/ IVERS,IUCALL,IUSED
+ /CONC/ NCONC
+ /DEVICE/ NREAD,NWRITE,NRDISK,NWDISK
+ /IDUM/ ILEN,JLEN
+ /LOGIC/ LDEN,LVIS,LTURB,LTEMP,LBUOY,LSCAL,LCOMP
+ ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
+ /MLTGRD/ MLEVEL,NLEVEL,ILEVEL
+ /SGLDBL/ IFLGPR,ICHKPR
+ /SPARM/ SMALL,SORMAX,NTTER,INDPRI,MAXIT,NODREF,NODMON
+ /TIMUSR/ DTUSR
+ /TRANSI/ NSTEP,KSTEP,MF,INCORE
+ /TRANSR/ TIME,DT,DTINV,TPARM
C

```

```

C+++++ USER AREA 2 ++++++
C---- AREA FOR USERS TO DECLARE THEIR OWN COMMON BLOCKS
C  THESE SHOULD START WITH THE CHARACTERS 'UC' TO ENSURE
C  NO CONFLICT WITH NON-USER COMMON BLOCKS
C
C+++++ END OF USER AREA 2 ++++++
C
  DIMENSION
+ U(NNODE,NPHASE),V(NNODE,NPHASE),W(NNODE,NPHASE),P(NNODE,NPHASE)
+,VFRAC(NNODE,NPHASE),DEN(NNODE,NPHASE),VIS(NNODE,NPHASE)
+,TE(NNODE,NPHASE),ED(NNODE,NPHASE),RS(NNODE,NPHASE,6)
+,T(NNODE,NPHASE),H(NNODE,NPHASE),RF(NNODE,NPHASE,4)
+,SCAL(NNODE,NPHASE,NSCAL)
  DIMENSION
+ XP(NNODE),YP(NNODE),ZP(NNODE)
+,VOL(NCELL),AREA(NFACE,3),VPOR(NCELL),ARPOR(NFACE,3)
+,WFACT(NFACE),CONV(NFACE,NPHASE)
+,IPT(*),IBLK(5,NBLOCK)
+,IPVERT(NCELL,8),IPNODN(NCELL,6),IPFACN(NCELL,6),IPNODF(NFACE,4)
+,IPNODB(NBDRY,4),IPFACB(NBDRY)
+,IWORK(*),WORK(*),CWORK(*)
  DIMENSION SGNWL(6)
C
C+++++ USER AREA 3 ++++++
C---- AREA FOR USERS TO DIMENSION THEIR ARRAYS
C
C---- AREA FOR USERS TO DEFINE DATA STATEMENTS
C
C+++++ END OF USER AREA 3 ++++++
C
  DATA SGNWL / 1.0, 1.0, 1.0, -1.0, -1.0, -1.0 /
C
C---- STATEMENT FUNCTION FOR ADDRESSING
C  IP(I,J,K)=IPT((K-1)*ILEN*JLEN+(J-1)*ILEN+I)
C
C----VERSION NUMBER OF USER ROUTINE AND PRECISION FLAG
C
  IVERS=3
  ICHKPR = 1
C
C+++++ USER AREA 4 ++++++
C---- TO USE THIS USER ROUTINE FIRST SET IUSED=1
C
  IUSED=1
C
C+++++ END OF USER AREA 4 ++++++
C
  IF (IUSED.EQ.0) RETURN
C
C---- FRONTEND CHECKING OF USER ROUTINE
  IF (IUCALL.EQ.0) RETURN
C
C+++++ USER AREA 5 ++++++
C  SET THE NUMBER OF PHASES
  IPHASE = 1
C
C  CALL SETWRK TO CREATE A WORKSPACE
  CALL SETWRK('USRTRN','WORK ',UGRAD,3*NCELL,JUGRAD)
  CALL SETWRK('USRTRN','WORK ',VGRAD,3*NCELL,JVGRAD)
  CALL SETWRK('USRTRN','WORK ',WGRAD,3*NCELL,JWGRAD)

```

```

C  CALL GRADV TO COMPUTE THE VELOCITY GRADIENTS
C  CALL GRADV('USRTRN',IPHASE,U(1,IPHASE),V(1,IPHASE),
+  W(1,IPHASE),WORK(JUGRAD),WORK(JVGRAD),WORK(JWGRAD),
+  XP,YP,ZP,VOL,AREA,IBLK,IPVERT,IPNODN,IPFACN,IPNODEF,
+  IPNODEB,IPFACB,WORK,IWORK,CWORK)

C  CALL IPALL TO FIND ADDRESSES
CALL IPALL(*,*,*,'BLOCK','CENTRES'
+  ,IPT,NPT,CWORK,IWORK)

C  LOOP OVER FLOW DOMAIN
DO 100 I=1,NPT
INODE=IPT(I)

C  SET USER SCALARS EQUAL TO THE NINE VELOCITY GRADIENTS

C  du/dx, du/dy, du/dz
SCAL(INODE,1,1)=WORK(JUGRAD+INODE-1)
SCAL(INODE,1,1)=WORK(JUGRAD+NCELL+INODE-1)
SCAL(INODE,1,3)=WORK(JUGRAD+2*NCELL+INODE-1)

C  dv/dx, dv/dy, dv/dz
SCAL(INODE,1,4)=WORK(JVGRAD+INODE-1)
SCAL(INODE,1,5)=WORK(JVGRAD+NCELL+INODE-1)
SCAL(INODE,1,6)=WORK(JVGRAD+2*NCELL+INODE-1)

C  dw/dx, dw/dy, dw/dz
SCAL(INODE,1,7)=WORK(JWGRAD+INODE-1)
SCAL(INODE,1,8)=WORK(JWGRAD+NCELL+INODE-1)
SCAL(INODE,1,9)=WORK(JWGRAD+2*NCELL+INODE-1)

100 CONTINUE

C  CALL DELWRK TO DELETE THE WORKSPACE
CALL DELWRK('USRSRC',WORK ,UGRAD )

C+++++ END OF USER AREA 5 ++++++
C
RETURN
END

```

A2 Subroutine USRTRN for the power dissipation function and the Movability Number

(Only user area 5 is given as routine is similar to A1)

```

C+++++++ USER AREA 5 ++++++
C
C
C  SET THE NUMBER OF PHASES
  IPHASE = 1

C  CALL SETWRK TO CREATE A WORKSPACE
  CALL SETWRK('USRTRN','WORK ',UGRAD,3*NCELL,JUGRAD)
  CALL SETWRK('USRTRN','WORK ',VGRAD,3*NCELL,JVGRAD)
  CALL SETWRK('USRTRN','WORK ',WGRAD,3*NCELL,JWGRAD)

C  CALL GRADV TO COMPUTE THE VELOCITY GRADIENTS
  CALL GRADV('USRTRN',IPHASE,U(1,IPHASE),V(1,IPHASE),
+  W(1,IPHASE),WORK(JUGRAD),WORK(JVGRAD),WORK(JWGRAD),
+  XP,YP,ZP,VOL,AREA,IBLK,IPVERT,IPNODN,IPFACN,IPNODF,
+  IPNODB,IPFACB,WORK,IWORK,CWORK)

C  CALL IPALL TO FIND ADDRESSES
  CALL IPALL(*,*, 'BLOCK', 'CENTRES'
+  ,IPT,NPT,CWORK,IWORK)

C  LOOP OVER FLOW DOMAIN
  DO 100 I=1,NPT
    INODE=IPT(I)

C  CALCULATION OF POWER DISSIPATION USING SHEAR STRESSES AND VELOCITY
    GRADIENTS

      SCAL(INODE,IPHASE,1)=+
C   $\frac{du}{dy} * \tau_{xy}$ 
+ABS(WORK(JUGRAD+NCELL+INODE-1)*RS(INODE,IPHASE,4))+
C   $\frac{du}{dz} * \tau_{xz}$ 
+ABS(WORK(JUGRAD+2*NCELL+INODE-1)*RS(INODE,IPHASE,6))+
C   $\frac{dv}{dx} * \tau_{xy}$ 
+ABS(WORK(JVGRAD+INODE-1)*RS(INODE,IPHASE,4))+
C   $\frac{dv}{dz} * \tau_{zy}$ 
+ABS(WORK(JVGRAD+2*NCELL+INODE-1)*RS(INODE,IPHASE,5))+
C   $\frac{dw}{dx} * \tau_{xz}$ 
+ABS(WORK(JWGRAD+INODE-1)*RS(INODE,IPHASE,6))+
C   $\frac{dw}{dy} * \tau_{zy}$ 
+ABS(WORK(JWGRAD+NCELL+INODE-1)*RS(INODE,IPHASE,5))+
C   $\frac{dw}{dz} * \tau_{zz}$ 
+ABS(WORK(JWGRAD+2*NCELL+INODE-1)*RS(INODE,IPHASE,3))+
C   $\frac{du}{dx} * \tau_{xx}$ 
+ABS(WORK(JUGRAD+INODE-1)*RS(INODE,IPHASE,1))+
C   $\frac{dv}{dy} * \tau_{yy}$ 
+ABS(WORK(JVGRAD+NCELL+INODE-1)*RS(INODE,IPHASE,2))

C  CALCULATION OF POWER DISSIPATION USING TOTAL VISCOSITY AND VELOCITY
    GRADIENTS

```

```

      SCAL(INODE,IPHASE,2)=VIS(INODE,IPHASE)*(2*+
C    $\frac{du}{dx}^2 \frac{dv}{dy}^2$ 
-   +((WORK(JUGRAD+INODE-1))**2+(WORK(JVGRAD+NCELL+INODE-1))**2+
C    $\frac{dw}{dz}^2 \frac{du}{dy}$ 
+   +(WORK(JWGRAD+2*NCELL+INODE-1))**2)+(WORK(JUGRAD+NCELL+INODE-1)+
C    $\frac{dv}{dx} \frac{dw}{dz}$ 
+   +WORK(JVGRAD+INODE-1))**2+(WORK(JUGRAD+2*NCELL+INODE-1)+
C    $\frac{dw}{dx} \frac{dv}{dz}$ 
+   +WORK(JWGRAD+INODE-1))**2+(WORK(JVGRAD+2*NCELL+INODE-1)+
C    $\frac{dw}{dy}$ 
+   +WORK(JWGRAD+NCELL+INODE-1))**2)

C   CALCULATION OF THE MOVABILITY NUMBER WHERE  $V_{ss} = 0.09\text{m/s}$ 
      SCAL(INODE,IPHASE,2)=(0.09**0.25)*(TE(INODE,IPHASE)**0.5)/0.09

100 CONTINUE

C   CALL DELWRK TO DELETE THE WORKSPACE
      CALL DELWRK('USRSRC',WORK ,'UGRAD ')
C
C+++++ END OF USER AREA 5 +++++

```

A3 Subroutine USRBCS for the inlet velocity distribution

```

SUBROUTINE USRBCS(VARBCS,VARAMB,A,B,C,ACND,BCND,CCND
+      ,IWGVEL,NDVWAL
+      ,FLOUT,NLABEL,NSTART,NEND,NCST,NCEN
+      ,U,V,W,P,VFRAC,DEN,VIS,TE,ED,RS,T,H,RF,SCAL
+      ,XP,YP,ZP,VOL,AREA,VPOR,ARPOR,WFACT,IPT
+      ,IBLK,IPVERT,IPNODN,IPFACN,IPNODF,IPNODB,IPFACB
+      ,WORK,IWORK,CWORK)
C
C *****
C
C USER ROUTINE TO SET REALS AT BOUNDARIES.
C
C >>> IMPORTANT <<<
C >>> <<<
C >>> USERS MAY ONLY ADD OR ALTER PARTS OF THE SUBROUTINE WITHIN <<<
C >>> THE DESIGNATED USER AREAS <<<
C
C *****
C
C THIS SUBROUTINE IS CALLED BY THE FOLLOWING SUBROUTINE
C CUSR SRLIST
C
C *****
C
C CREATED
C 30/11/88 ADB
C MODIFIED
C 08/09/90 ADB RESTRUCTURED FOR USER-FRIENDLINESS.
C 10/08/91 IRH FURTHER RESTRUCTURING ADD ACND BCND CCND
C 22/09/91 IRH CHANGE ICALL TO IUCALL + ADD /SPARM/
C 10/03/92 PHA UPDATE CALLED BY COMMENT, ADD RF ARGUMENT,
C CHANGE LAST DIMENSION OF RS TO 6 AND IVERS TO 2
C 03/06/92 PHA ADD PRECISION FLAG AND CHANGE IVERS TO 3
C 30/06/92 NSW INCLUDE FLAG FOR CALLING BY ITERATION
C INSERT EXTRA COMMENTS
C 03/08/92 NSW MODIFY DIMENSION STATEMENTS FOR VAX
C 21/12/92 CSH INCREASE IVERS TO 4
C 02/08/93 NSW INCORRECT AND MISLEADING COMMENT REMOVED
C 05/11/93 NSW INDICATE USE OF FLOUT IN MULTIPHASE FLOWS
C 23/11/93 CSH EXPLICITLY DIMENSION IPVERT ETC.
C 01/02/94 NSW SET VARIABLE POINTERS IN WALL EXAMPLE.
C CHANGE FLOW3D TO CFDS-FLOW3D.
C MODIFY MULTIPHASE MASS FLOW BOUNDARY TREATMENT.
C 03/03/94 FHW CORRECTION OF SPELLING MISTAKE
C 02/07/94 BAS SLIDING GRIDS - ADD NEW ARGUMENT IWGVEL
C TO ALLOW VARIANTS OF TRANSIENT-GRID WALL BC
C CHANGE VERSION NUMBER TO 5
C 09/08/94 NSW CORRECT SPELLING
C MOVE 'IF(USED.EQ.0) RETURN' OUT OF USER AREA
C 19/12/94 NSW CHANGE FOR CFX-F3D
C 02/02/95 NSW CHANGE COMMON /IMFBMP/
C 02/06/97 NSW MAKE EXAMPLE MORE LOGICAL
C 02/07/97 NSW UPDATE FOR CFX-4
C 08/09/98 NSW CORRECT SIZE OF WALL ARRAY IN COMMENT
C
C *****
C
C SUBROUTINE ARGUMENTS

```

C
 C VARBCS - REAL BOUNDARY CONDITIONS
 C VARAMB - AMBIENT VALUE OF VARIABLES
 C A - COEFFICIENT IN WALL BOUNDARY CONDITION
 C B - COEFFICIENT IN WALL BOUNDARY CONDITION
 C C - COEFFICIENT IN WALL BOUNDARY CONDITION
 C ACND - COEFFICIENT IN CONDUCTING WALL BOUNDARY CONDITION
 C BCND - COEFFICIENT IN CONDUCTING WALL BOUNDARY CONDITION
 C CCND - COEFFICIENT IN CONDUCTING WALL BOUNDARY CONDITION
 C IWGVEL - USAGE OF INPUT VELOCITIES (0 = AS IS, 1 = ADD GRID MOTION)
 C NDVWAL - FIRST DIMENSION OF ARRAY IWGVEL
 C FLOUT - MASS FLOW/FRACTIONAL MASS FLOW
 C NLABEL - NUMBER OF DISTINCT OUTLETS
 C NSTART - ARRAY POINTER
 C NEND - ARRAY POINTER
 C NCST - ARRAY POINTER
 C NCEN - ARRAY POINTER
 C U - U COMPONENT OF VELOCITY
 C V - V COMPONENT OF VELOCITY
 C W - W COMPONENT OF VELOCITY
 C P - PRESSURE
 C VFRAC - VOLUME FRACTION
 C DEN - DENSITY OF FLUID
 C VIS - VISCOSITY OF FLUID
 C TE - TURBULENT KINETIC ENERGY
 C ED - EPSILON
 C RS - REYNOLD STRESSES
 C T - TEMPERATURE
 C H - ENTHALPY
 C RF - REYNOLD FLUXES
 C SCAL - SCALARS (THE FIRST 'NCONC' OF THESE ARE MASS FRACTIONS)
 C XP - X COORDINATES OF CELL CENTRES
 C YP - Y COORDINATES OF CELL CENTRES
 C ZP - Z COORDINATES OF CELL CENTRES
 C VOL - VOLUME OF CELLS
 C AREA - AREA OF CELLS
 C VPOR - POROUS VOLUME
 C ARPOR - POROUS AREA
 C WFACT - WEIGHT FACTORS
 C
 C IPT - 1D POINTER ARRAY
 C IBLK - BLOCK SIZE INFORMATION
 C IPVERT - POINTER FROM CELL CENTERS TO 8 NEIGHBOURING VERTICES
 C IPNODN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING CELLS
 C IPFACN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING FACES
 C IPNODF - POINTER FROM CELL FACES TO 2 NEIGHBOURING CELL CENTERS
 C IPNODB - POINTER FROM BOUNDARY CENTERS TO CELL CENTERS
 C IPFACB - POINTER TO NODES FROM BOUNDARY FACES
 C
 C WORK - REAL WORKSPACE ARRAY
 C IWORK - INTEGER WORKSPACE ARRAY
 C CWORK - CHARACTER WORKSPACE ARRAY
 C
 C SUBROUTINE ARGUMENTS PRECEDED WITH A '*' ARE ARGUMENTS THAT MUST
 C BE SET BY THE USER IN THIS ROUTINE.
 C
 C NOTE THAT OTHER DATA MAY BE OBTAINED FROM CFX-4 USING THE
 C ROUTINE GETADD, FOR FURTHER DETAILS SEE THE VERSION 4
 C USER MANUAL.
 C

```

      ELSE
      VARBCS(IU,IPHS,INODE)=LOG(YP(INODE)/0.001)/
- + LOG(0.043/(EXP(1.)*0.001))*
+ LOG(ABS(0.1-ZP(INODE))/0.001)/
+ LOG(0.05/(EXP(1.)*0.001))*
+ 0.372
      END IF
101  CONTINUE
102  CONTINUE
103  CONTINUE
C
C+++++ END OF USER AREA 5 +++++
C
      RETURN
      END

```

```

C*****
  LOGICAL LDEN,LVIS,LTURB,ITEMP,LBUOY,LSCAL,LCOMP
- + ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
C
  CHARACTER*(*) CWORK
C
C+++++ USER AREA 1 ++++++
C---- AREA FOR USERS EXPLICITLY DECLARED VARIABLES
C
C+++++ END OF USER AREA 1 ++++++
C
  COMMON
+ /ALL/  NBLOCK,NCELL,NBDRY,NNODE,NFACE,NVERT,NDIM
+ /ALLWRK/ NRWS,NIWS,NCWS,IWRFRE,IWIFRE,IWCFRE
+ /ADDIMS/ NPHASE,NSCAL,NVAR,NPROP
+ ,NDVAR,NDPROP,NDXNN,NDGEOM,NDCOE,NLIST,NRLIST,NTOPOL
+ /BCSOUT/ IFLOUT
+ /CHKUSR/ IVERS,IUCALL,IUSED
+ /DEVICE/ NREAD,NWRITE,NRDISK,NWDISK
+ /IDUM/  ILEN,JLEN
+ /IMFBMP/ IMFBMP,JMFBMP
+ /LOGIC/ LDEN,LVIS,LTURB,ITEMP,LBUOY,LSCAL,LCOMP
+ ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
+ /MLTGRD/ MLEVEL,NLEVEL,ILEVEL
+ /SGLDBL/ IFLGPR,ICHKPR
+ /SPARM/  SMALL,SORMAX,NITER,INDPRI,MAXIT,NODREF,NODMON
+ /TRANSI/ NSTEP,KSTEP,MF,INCORE
+ /TRANSR/ TIME,DT,DTINV,TPARM
+ /UBCSFL/ IUBCSF
C
C+++++ USER AREA 2 ++++++
C---- AREA FOR USERS TO DECLARE THEIR OWN COMMON BLOCKS
C  THESE SHOULD START WITH THE CHARACTERS 'UC' TO ENSURE
C  NO CONFLICT WITH NON-USER COMMON BLOCKS
C
C+++++ END OF USER AREA 2 ++++++
C
  DIMENSION
+ VARBCS(NVAR,NPHASE,NCELL+1:NNODE),VARAMB(NVAR,NPHASE)
+,A(4+NSCAL,NPHASE,NSTART:*)
+,B(4+NSCAL,NPHASE,NSTART:*),C(4+NSCAL,NPHASE,NSTART:*)
+,FLOUT(*),ACND(NCST:*),BCND(NCST:*),CCND(NCST:*)
+,IWGVEL(NDVWAL,NPHASE)
C
  DIMENSION
+ U(NNODE,NPHASE),V(NNODE,NPHASE),W(NNODE,NPHASE),P(NNODE,NPHASE)
+,VFRAC(NNODE,NPHASE),DEN(NNODE,NPHASE),VIS(NNODE,NPHASE)
+,TE(NNODE,NPHASE),ED(NNODE,NPHASE),RS(NNODE,NPHASE,6)
+,T(NNODE,NPHASE),H(NNODE,NPHASE),RF(NNODE,NPHASE,4)
+,SCAL(NNODE,NPHASE,NSCAL)
C
  DIMENSION
+ XP(NNODE),YP(NNODE),ZP(NNODE)
+,VOL(NCELL),AREA(NFACE,3),VPOR(NCELL),ARPOR(NFACE,3),WFACT(NFACE)
+,IPT(*),IBLK(5,NBLOCK)
+,IPVERT(NCELL,8),IPNODN(NCELL,6),IPFACN(NCELL,6),IPNODF(NFACE,4)
+,IPNODB(NBDRY,4),IPFACB(NBDRY)
+,IWORK(*),WORK(*),CWORK(*)
C
C+++++ USER AREA 3 ++++++

```

```

C--- AREA FOR USERS TO DIMENSION THEIR ARRAYS
C
C--- AREA FOR USERS TO DEFINE DATA STATEMENTS
C
C+++++ END OF USER AREA 3 ++++++
C
C--- STATEMENT FUNCTION FOR ADDRESSING
      IP(I,J,K)=IPT((K-1)*ILEN*JLEN+(J-1)*ILEN+I)
C
C---VERSION NUMBER OF USER ROUTINE AND PRECISION FLAG
C
      IVERS=5
      ICHKPR = 1
C
C+++++ USER AREA 4 ++++++
C--- TO USE THIS USER ROUTINE FIRST SET IUSED=1
C   AND SET IUBCSF FLAG:
C   BOUNDARY CONDITIONS NOT CHANGING           IUBCSF=0
C   BOUNDARY CONDITIONS CHANGING WITH ITERATION IUBCSF=1
C   BOUNDARY CONDITIONS CHANGING WITH TIME      IUBCSF=2
C   BOUNDARY CONDITIONS CHANGING WITH TIME AND ITERATION IUBCSF=3
      IUBCSF=0
      IUSED=1
C
C+++++ END OF USER AREA 4 ++++++
C
      IF (IUSED.EQ.0) RETURN
C
C--- FRONTEND CHECKING OF USER ROUTINE
      IF (IUCALL.EQ.0) RETURN
C
C+++++ USER AREA 5 ++++++
C
C-- INTERROGATE GETVAR FOR VARIABLE NUMBERS I.E. U-VELOCITY
C
      CALL GETVAR('USRBCS','U',IU)
C
C
C SET IPHS = 1 FOR SINGLE PHASE FLOW.
C
      IPHS = 1
C
C CALL IPREC TO FIND ADDRESSES
      CALL IPREC('INLET','PATCH','CENTRES',IPT,ILEN,JLEN,KLEN,
+      CWORK,IWORK)
C
C LOOP OVER INLET PATCH
      DO 103 K = 1, KLEN
        DO 102 J = 1, JLEN
          DO 101 I = 1, ILEN
C USE STATEMENT FUNCTION IP TO GET ADDRESSES
          INODE = IP(I,J,K)
C CALCULATE LOGARITHMIC VELOCITY DISTRIBUTION
C DEPTH  $y = 43\text{mm}$ , WIDTH  $B = 0.1\text{mm}$ ,  $y_o = 0.001\text{mm}$ , AVERAGE VELOCITY  $\bar{u} = 0.372\text{m/s}$ 
          IF (ZP(INODE)<=0.05) THEN
            VARBCS(IU,IPHS,INODE)=LOG(YP(INODE)/0.001)/
+      LOG(0.043/(EXP(1.)*0.001))*
+      LOG(ABS(ZP(INODE)/0.001))/
+      LOG(0.05/(EXP(1.)*0.001))*
+      0.372

```

A4 Subroutine USRWTM for the incorporating roughness effects

```

SUBROUTINE USRWTM(CNAME,IEQN,IPHASE,CTROP,CWALL
+      ,GAMMA,YWALL,TMULT,ISTART,NCV
+      ,DIFLAM,XYPLUS,ELOG,PRT
+      ,U,V,W,P,VFRAC,DEN,VIS,TE,ED,RS,T,H,RF,SCAL
+      ,XP,YP,ZP,VOL,AREA,VPOR,ARPOR,WFACT,IPT
+      ,IBLK,IPVERT,IPNODN,IPFACN,IPNODF,IPNODB,IPFACB
+      ,WORK,IWORK,CWORK)
C
C *****
C
C USER SUBROUTINE TO OVERWRITE DEFAULT WALL MULTIPLIERS (TMULT)
C FLUX = TMULT * (VARIABLE AT WALL - VARIABLE AT NEAREST NODE)
C
C >>> IMPORTANT <<<
C >>> <<<
C >>> USERS MAY ONLY ADD OR ALTER PARTS OF THE SUBROUTINE WITHIN <<<
C >>> THE DESIGNATED USER AREAS <<<
C
C *****
C
C THIS SUBROUTINE IS CALLED BY THE FOLLOWING SUBROUTINE
C CUSR WALTML
C
C *****
C CREATED
C 03/11/93 NSW
C MODIFIED
C 03/02/94 PHA CHANGE FLOW3D TO CFDS-FLOW3D
C 09/08/94 NSW CORRECT SPELLING
C MOVE IF(USED.EQ.0) RETURN OUT OF USER AREA
C 16/11/94 NSW AVOID USE OF YP AS SCALAR IN EXAMPLE.
C INCLUDE NEW EXAMPLE FOR SMOOTH WALL WITH
C VARIABLE MOLECULAR VISCOSITY.
C 19/12/94 NSW CHANGE FOR CFX-F3D
C 11/01/95 NSW CORRECT NEW EXAMPLE
C 15/04/96 NSW INCLUDE HEAT TRANSFER COEFFICIENT EXAMPLE
C 02/07/97 NSW UPDATE FOR CFX-4
C 25/08/98 NSW CORRECT LOGIC OF EXAMPLES
C 24/03/99 NSW CHANGE RECOMMENDED FORMULA FOR ROUGH WALLS
C
C *****
C
C SUBROUTINE ARGUMENTS
C
C CNAME - EQUATION NAME
C IEQN - EQUATION NUMBER
C IPHASE - PHASE NUMBER
C CTROP - ISOTROPIC/ANISORTROPIC FLAG
C CWALL - NATURAL/SYNTHETIC FLAG
C GAMMA - LAMINAR DIFFUSIVITY
C YWALL - DISTANCE OF NODE FROM WALL
C * TMULT - MULTIPLIER TO BE SET WHERE REQUIRED
C ISTART - FIRST BOUNDARY NODE ON WALL
C NCV - NUMBER OF WALL NODES
C DIFLAM - CONSTANT LAMINAR DIFFUSIVITY
C XYPLUS - THICKNESS OF LAMINAR SUBLAYER
C ELOG - LOGLAYER MULTIPLIER

```

```

C  PRT  - PRANDTL NUMBER
C  U    - U COMPONENT OF VELOCITY
C  V    - V COMPONENT OF VELOCITY
C  W    - W COMPONENT OF VELOCITY
C  P    - PRESSURE
C  VFRAC - VOLUME FRACTION
C  DEN  - DENSITY OF FLUID
C  VIS  - VISCOSITY OF FLUID
C  TE   - TURBULENT KINETIC ENERGY
C  ED   - EPSILON
C  RS   - REYNOLDS STRESSES
C  T    - TEMPERATURE
C  H    - ENTHALPY
C  RF   - REYNOLDS FLUXES
C  SCAL - SCALARS (THE FIRST 'NCONC' OF THESE ARE MASS FRACTIONS)
C  XP   - X COORDINATES OF CELL CENTRES
C  YP   - Y COORDINATES OF CELL CENTRES
C  ZP   - Z COORDINATES OF CELL CENTRES
C  VOL  - VOLUME OF CELLS
C  AREA - AREA OF CELLS
C  VPOR - POROUS VOLUME
C  ARPOR - POROUS AREA
C  WFACT - WEIGHT FACTORS
C
C  IPT  - 1D POINTER ARRAY
C  IBLK - BLOCK SIZE INFORMATION
C  IPVERT - POINTER FROM CELL CENTERS TO 8 NEIGHBOURING VERTICES
C  IPNODN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING CELLS
C  IPFACN - POINTER FROM CELL CENTERS TO 6 NEIGHBOURING FACES
C  IPNODF - POINTER FROM CELL FACES TO 2 NEIGHBOURING CELL CENTERS
C  IPNODB - POINTER FROM BOUNDARY CENTERS TO CELL CENTERS
C  IPFACB - POINTER FROM BOUNDARY CENTERS TO BOUNDARY FACES
C
C  WORK - REAL WORKSPACE ARRAY
C  IWORK - INTEGER WORKSPACE ARRAY
C  CWORK - CHARACTER WORKSPACE ARRAY
C
C  SUBROUTINE ARGUMENTS PRECEDED WITH A '*' ARE ARGUMENTS THAT MUST
C  BE SET BY THE USER IN THIS ROUTINE.
C
C  NOTE THAT OTHER DATA MAY BE OBTAINED FROM CFX-4 USING THE
C  ROUTINE GETADD, FOR FURTHER DETAILS SEE THE VERSION 4
C  USER MANUAL.
C
C *****
C
C  LOGICAL LDEN,LVIS,LTURB,LTEMP,LBUOY,LSCAL,LCOMP
C  + ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
C
C  CHARACTER*(*) CWALL,CWORK
C  CHARACTER  CNAME*6,CTROP*6
C
C  ++++++ USER AREA 1 ++++++
C  ---- AREA FOR USERS EXPLICITLY DECLARED VARIABLES
C
C  ++++++ END OF USER AREA 1 ++++++
C
C  COMMON
C  + /ALL/  NBLOCK,NCELL,NBDRY,NNODE,NFACE,NVERT,NDIM
C  + /ADDIMS/ NPHASE,NSCAL,NVAR,NPROP

```

```

+ ,NDVAR,NDPROP,NDXNN,NDGEOM,NDCOEF,NILIST,NRLIST,NTOPOL
+ /CHKUSR/ IVERS,IUCALL,IUSED
- + /DEVICE/ NREAD,NWRITE,NRDISK,NWDISK
+ /IDUM/ ILEN,JLEN
+ /LOGIC/ LDEN,LVIS,LTURB,LTEMP,LBOUY,LSCAL,LCOMP
+ ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
+ /MLTGRD/ MLEVEL,NLEVEL,ILEVEL
+ /SGLDBL/ IFLGPR,ICHKPR
+ /SPARM/ SMALL,SORMAX,NITER,INDPRI,MAXIT,NODREF,NODMON
+ /TRANS/ NSTEP,KSTEP,MF,INCORE
+ /TRANSR/ TIME,DT,DTINV,TPARM
+ /TURBKE/ CMU,C1,C2,C3,CAPPA,CMU14,CMU34
C
C+++++ USER AREA 2 ++++++
C--- AREA FOR USERS TO DECLARE THEIR OWN COMMON BLOCKS
C  THESE SHOULD START WITH THE CHARACTERS 'UC' TO ENSURE
C  NO CONFLICT WITH NON-USER COMMON BLOCKS
C
C+++++ END OF USER AREA 2 ++++++
C
  DIMENSION CWALL(NPHASE),GAMMA(NCELL,NPHASE,*)
  +,YWALL(NCV),TMULT(NCV,NPHASE)
  +,DIFLAM(NVAR,NPHASE),XYPLUS(NVAR,NPHASE),ELOG(NVAR,NPHASE)
  +,PRT(NVAR,NPHASE)
  DIMENSION
  + U(NNODE,NPHASE),V(NNODE,NPHASE),W(NNODE,NPHASE),P(NNODE,NPHASE)
  +,VFRAC(NNODE,NPHASE),DEN(NNODE,NPHASE),VIS(NNODE,NPHASE)
  +,TE(NNODE,NPHASE),ED(NNODE,NPHASE),RS(NNODE,NPHASE,6)
  +,T(NNODE,NPHASE),H(NNODE,NPHASE),RF(NNODE,NPHASE,4)
  +,SCAL(NNODE,NPHASE,NSCAL)
  DIMENSION
  + XP(NNODE),YP(NNODE),ZP(NNODE)
  +,VOL(NCELL),AREA(NFACE,3),VPOR(NCELL),ARPOR(NFACE,3),WFACT(NFACE)
  +,IPT(*),IBLK(5,NBLOCK)
  +,IPVERT(NCELL,8),IPNODN(NCELL,6),IPFACN(NCELL,6),IPNODF(NFACE,4)
  +,IPNODB(NBDRY,4),IPFACB(NBDRY)
  +,IWORK(*),WORK(*),CWORK(*)
C
C+++++ USER AREA 3 ++++++
C--- AREA FOR USERS TO DIMENSION THEIR ARRAYS
C
C--- AREA FOR USERS TO DEFINE DATA STATEMENTS
C
C+++++ END OF USER AREA 3 ++++++
C
C--- STATEMENT FUNCTION FOR ADDRESSING
  IP(I,J,K)=IPT((K-1)*ILEN*JLEN+(J-1)*ILEN+I)
C
C--- VERSION NUMBER OF USER ROUTINE AND PRECISION FLAG
C
  IVERS = 1
  ICHKPR = 1
C
C+++++ USER AREA 4 ++++++
C--- TO USE THIS USER ROUTINE FIRST SET IUSED=1
C
  IUSED = 1
C
C+++++ END OF USER AREA 4 ++++++
C

```

```

      IF (IUSED.EQ.0) RETURN
C
C----- FRONTEND CHECKING OF USER ROUTINE
      IF (IUCALL.EQ.0) RETURN
C
C+++++++ USER AREA 5 ++++++++
C      CALL THE PATCH AT THE CHANNEL BED USING IPREC
      CALL IPREC('WALL_BOTTOM','PATCH','CENTRES',IPT,ILEN,JLEN,KLEN,
+      CWORK,IWORK)

C      SET THE ROUGHNESS FACTOR AT 0.3
      RFACT=0.3

C      SET THE ROUGHNESS HEIGHT e.g. 86mm PARTICLE SIZE
      RHT=0.00086

C      INTERROGATE GETVAR FOR THE VELOCITY COMPONENTS
      CALL GETVAR('USRWTM','U',IVAR)

C      CALCULATE NEW VELOCITY PROFILE (CFX Manual 1999)
      IF (IVAR.EQ.IEQN) THEN
        VISCOS = DIFLAM(IVAR,1)
        ELOGS = ELOG(IVAR,1)
        PRIT = PRT(IVAR,1)
        FACT = CMU14/VISCOS
        DO 113 K = 1, KLEN
          DO 112 J = 1, JLEN
            DO 111 I = 1, ILEN
              INODE = IP(I,J,K)
              IBDY = INODE - NCELL
              INODE1 = IPNODB(IBDY,1)
              LCV = IBDY - ISTART + 1
              DENSQK = DEN(INODE1,1) * SQRT( TE(INODE1,1) )
              YPLUS = FACT*DENSQK*YWALL(LCV)
              HPLUS = FACT*DENSQK*RHT
              ANUM = PRIT*LOG(ELOGS*YPLUS/(1.0+RFACT*HPLUS)) / CAPPA
              TMULT(LCV,1) = DENSQK*CMU14 / ANUM
111          CONTINUE
112        CONTINUE
113      CONTINUE
          END IF
C+++++++ END OF USER AREA 5 ++++++++
C
      RETURN
      END

```

A5 Subroutine USRBCS for setting a moving wall

(Only user area 5 and 6 are given as routine is similar to A3)

```

C+++++ USER AREA 5 ++++++
C-- INTERROGATE GETVAR FOR VARIABLE NUMBERS.
C
  CALL GETVAR('USRBCS','U ',IU)
C
C+++++ END OF USER AREA 5 ++++++
C
C+++++ USER AREA 6 ++++++

C-- SET IPHS = 1 FOR SINGLE PHASE FLOW.
C
  IPHS = 1
C
C USE IPREC TO FIND 1D ADDRESSES OF A GROUP OF PATCH CENTRES
C
  CALL IPREC('WALL_TOP','PATCH','CENTRES',IPT,ILEN,JLEN,KLEN,
+          CWORK,IWORK)

  DO 113 K = 1, KLEN
    DO 112 J = 1, JLEN
      DO 111 I = 1, ILEN
C   SET A=1 I.E. VELOCITY BEING SET
      A(IU,IPHS,INODE) = 1.0
C   SET B=0 I.E. SHEAR STRESS ZERO AT WALL
      B(IU,IPHS,INODE) = 0.0
C   SET C AS THE VELOCITY FUNCTION I.E. LOGARITHMIC PROFILE
      IF (ZP(INODE)<=0.05) THEN
        C(IU,IPHS,INODE)=LOG(YP(INODE)/0.0001)/
+      LOG(0.043/(EXP(1.)*0.001))*
+      LOG(ABS(ZP(INODE))/0.001)/
+      LOG(0.05/(EXP(1.)*0.001))*
+      0.372
      ELSE
        C(IU,IPHS,INODE)=LOG(YP(INODE)/0.001)/
+      LOG(0.043/(EXP(1.)*0.001))*
+      LOG(ABS(0.1-ZP(INODE))/0.001)/
+      LOG(0.05/(EXP(1.)*0.001))*
+      0.372
      END IF
C   A(IV,IPHS,INODE) = 0.0
C   B(IV,IPHS,INODE) = 1.0
C   C(IV,IPHS,INODE) = 0.0

C   A(TW,IPHS,INODE) = 0.0
C   B(TW,IPHS,INODE) = 1.0
C   C(TW,IPHS,INODE) = 0.0
111  CONTINUE
112  CONTINUE
113  CONTINUE
C+++++ END OF USER AREA 6 ++++++

```

A6 Subroutine USRBCS for calculating k and ε distributions

(Only user area 5 is given as routine is similar to A3)

```

C+++++++ USER AREA 5 ++++++
C
C SET IPHS = 1 FOR SINGLE PHASE FLOW.
C
  IPHS = 1
C INTERROGATE GETVAR FOR THE VELOCITY, TURBULENT KINETIC ENERGY AND
  RATE OF VISCOUS DISSIPATION ARRAY
  CALL GETVAR('USRBCS','U','IU)
  CALL GETVAR('USRBCS','TE','IK)
  CALL GETVAR('USRBCS','ED','IE)

C USE IPREC TO GET CELL ADDRESSES OF INLET PATCH
  CALL IPREC('INLET','PATCH','CENTRES',IPT,ILEN,JLEN,KLEN,
+    CWORK,IWORK)

C LOOP OVER INLET PATCH
  DO 103 K = 1, KLEN
    DO 102 J = 1, JLEN
      DO 101 I = 1, ILEN
        INODE = IP(I,J,K)
C E.G. OF ONE-DIMENSIONAL INLET VELOCITY DISTRIBUTION
        VARBCS(IU,NPHASE,INODE)=(log(YP(INODE)/1.0135E-7)/log(0.1495/
+          (exp(1.)*1.0135E-7)))*0.1646348

C COMPUTE THE TURBULENT KINETIC ENERGY DISTRIBUTION
        VARBCS(IK,NPHASE,INODE)=-277.68*((YP(INODE))**6)
+          +131.05*((YP(INODE))**5)
+          -22.723*((YP(INODE))**4)
+          +1.8371*((YP(INODE))**3)
+          -0.072*((YP(INODE))**2)
+          -0.0004*YP(INODE)+0.0002

C CALCULATE THE RATE OF VISCOUS DISSIPATION DISTRIBUTION
        VARBCS(IE,NPHASE,INODE)=3008.1*((YP(INODE))**6)
+          -1544. *((YP(INODE))**5)
+          +312.86*((YP(INODE))**4)
+          -31.727*((YP(INODE))**3)
+          +1.6824*((YP(INODE))**2)
+          -0.0443*YP(INODE)+0.0005
101 CONTINUE
102 CONTINUE
103 CONTINUE

C+++++++ END OF USER AREA 5 ++++++

```

A7 Subroutine USRTRN for calculating the bedform critical Function

(Only user areas 1-5 ARE given as routine is similar to A1)

```

C+++++ USER AREA 1 ++++++
C--- AREA FOR USERS EXPLICITLY DECLARED VARIABLES
C  DECLARE VARIABLES
    INTEGER ios,out,out1
C+++++ END OF USER AREA 1 ++++++
C
  COMMON
+ /ALL/  NBLOCK,NCELL,NBDRY,NNODE,NFACE,NVERT,NDIM
+ /ALLWRK/ NRWS,NIWS,NCWS,IWRFRE,IWIFRE,IWCFRE
+ /ADDIMS/ NPHASE,NSCAL,NVAR,NPROP
+         ,NDVAR,NDPROP,NDXNN,NDGEOM,NDCOEF,NILIST,NRLIST,NTOPOL
+ /CHKUSR/ IVERS,IUCALL,IUSED
+ /CONC/  NCONC
+ /DEVICE/ NREAD,NWRITE,NRDISK,NWDISK
+ /IDUM/  ILEN,JLEN
+ /LOGIC/ LDEN,LVIS,LTURB,LTEMP,LBUOY,LSCAL,LCOMP
+         ,LRECT,LCYN,LAXIS,LPOROS,LTRANS
+ /MLTGRD/ MLEVEL,NLEVEL,ILEVEL
+ /SGLDBL/ IFLGPR,ICHKPR
+ /SPARM/  SMALL,SORMAX,NITER,INDPRI,MAXIT,NODREF,NODMON
+ /TIMUSR/ DTUSR
+ /TRANSI/ NSTEP,KSTEP,MF,INCORE
+ /TRANSR/ TIME,DT,DTINVF,TPARM
C
C+++++ USER AREA 2 ++++++
C--- AREA FOR USERS TO DECLARE THEIR OWN COMMON BLOCKS
C  THESE SHOULD START WITH THE CHARACTERS 'UC' TO ENSURE
C  NO CONFLICT WITH NON-USER COMMON BLOCKS
C
C+++++ END OF USER AREA 2 ++++++
C
  DIMENSION
+ U(NNODE,NPHASE),V(NNODE,NPHASE),W(NNODE,NPHASE),P(NNODE,NPHASE)
+ ,VFRAC(NNODE,NPHASE),DEN(NNODE,NPHASE),VIS(NNODE,NPHASE)
+ ,TE(NNODE,NPHASE),ED(NNODE,NPHASE),RS(NNODE,NPHASE,6)
+ ,T(NNODE,NPHASE),H(NNODE,NPHASE),RF(NNODE,NPHASE,4)
+ ,SCAL(NNODE,NPHASE,NSCAL)
  DIMENSION
+ XP(NNODE),YP(NNODE),ZP(NNODE)
+ ,VOL(NCELL),AREA(NFACE,3),VPOR(NCELL),ARPOR(NFACE,3)
+ ,WFACT(NFACE),CONV(NFACE,NPHASE)
+ ,IPT(*),IBLK(5,NBLOCK)
+ ,IPVERT(NCELL,8),IPNODN(NCELL,6),IPFACN(NCELL,6),IPNODF(NFACE,4)
+ ,IPNODB(NBDRY,4),IPFACB(NBDRY)
+ ,IWORK(*),WORK(*),CWORK(*)
  DIMENSION SGNWL(6)
C
C+++++ USER AREA 3 ++++++
C--- AREA FOR USERS TO DIMENSION THEIR ARRAYS
C  DIMENSION
C  +NORMX(NNODE,NPHASE),NORMY(NNODE,NPHASE),NORMZ(NNODE,NPHASE)
C--- AREA FOR USERS TO DEFINE DATA STATEMENTS

```

```

C+++++ END OF USER AREA 3 ++++++
C
- DATA SGNWL / 1.0, 1.0, 1.0, -1.0, -1.0, -1.0 /
C
C--- STATEMENT FUNCTION FOR ADDRESSING
C IP(I,J,K)=IPT((K-1)*ILEN*JLEN+(J-1)*ILEN+I)
C
C--- VERSION NUMBER OF USER ROUTINE AND PRECISION FLAG
C
  IVERS=3
  ICHKPR = 1
C
C+++++ USER AREA 4 ++++++
C--- TO USE THIS USER ROUTINE FIRST SET IUSED=1
C
  IUSED=1
C          SET FILE NUMBER AND ERROR STATUS
  out=10
  ios=0

C+++++ END OF USER AREA 4 ++++++
C
  IF (IUSED.EQ.0) RETURN
C
C--- FRONTEND CHECKING OF USER ROUTINE
  IF (IUCALL.EQ.0) RETURN
C
C+++++ USER AREA 5 ++++++
C
C  SET IPHASE = 1 FOR SINGLE PHASE FLOW
  IPHASE = 1

C  USE IPALL TO ACCESS THE WALL BOUNDARY PATCH AT CHANNEL BED
  CALL IPALL('WALL_BOTTOM','WALL','PATCH','CENTRES'
+           ,IPT,NPT,CWORK,IWORK)

C  LOOP THROUGH PATCH
  DO 100 I=1,NPT
    INODE=IPT(I)
    IBDRY=INODE-NCCELL
    IFACE=IPFACB(IBDRY)

C  NORMAL DIRECTION VECTOR (X,Y,Z) GIVEN BY (SCAL1, SCAL2, SCAL3)
  SCAL(I,IPHASE,1)=AREA(IFACE,1)
  SCAL(I,IPHASE,2)=AREA(IFACE,2)
  SCAL(I,IPHASE,3)=AREA(IFACE,3)

  100 CONTINUE

C  USE IPALL TO ACCESS THE USER3D PATCH AT CHANNEL BED
  CALL IPALL('USER3D_B','USER3D','PATCH','CENTRES'
+           ,IPT,NPT,CWORK,IWORK)

C  LOOP THROUGH PATCH
  DO 200 I=1,NPT
    INODE=IPT(I)
    IBDRY=INODE-NCCELL

C  t- VECTOR (X,Y,Z) GIVEN BY (SCAL4, SCAL5, SCAL 6)
    SCAL(INODE,IPHASE,4)= V(INODE,IPHASE)*SCAL(I,IPHASE,3)-

```

```

+      W(INODE,IPHASE)*SCAL(I,IPHASE,2)
SCAL(INODE,IPHASE,5)= W(INODE,IPHASE)*SCAL(I,IPHASE,1)-
- +      - U(INODE,IPHASE)*SCAL(I,IPHASE,3)
SCAL(INODE,IPHASE,6)= U(INODE,IPHASE)*SCAL(I,IPHASE,2)-
+      V(INODE,IPHASE)*SCAL(I,IPHASE,1)

```

C COMPUTE TAN BETA

```

      IF (((W(INODE,IPHASE)>0.1E-12).OR.(W(INODE,IPHASE)<-0.1E-12)).AND.
+ ((U(INODE,IPHASE)>0.1E-12).OR.(U(INODE,IPHASE)<-0.1E-12)))THEN
      SCAL(INODE,IPHASE,7)=-V(INODE,IPHASE)/((U(INODE,IPHASE)**2.+
+      W(INODE,IPHASE)**2.)*0.5)
      IF (((W(INODE,IPHASE)>0.1E-12).OR.(W(INODE,IPHASE)<-0.1E-12))
+ .AND.((U(INODE,IPHASE)<0.1E-12).AND.(U(INODE,IPHASE)>-0.1E-12)))
+ THEN
      SCAL(INODE,IPHASE,7)=-V(INODE,IPHASE)/ABS(W(INODE,IPHASE))
      IF (((W(INODE,IPHASE)<0.1E-12).AND.(W(INODE,IPHASE)>-0.1E-12))
+ .AND.((U(INODE,IPHASE)>0.1E-12).OR.(U(INODE,IPHASE)<-0.1E-12)))
+ THEN
      SCAL(INODE,IPHASE,7)=-V(INODE,IPHASE)/ABS(U(INODE,IPHASE))
      ELSE
      SCAL(INODE,IPHASE,7)=-V(INODE,IPHASE)
      END IF
END IF
      END IF

```

C COMPUTE TAN GAMMA

```

      IF (((SCAL(INODE,IPHASE,6)>0.1E-12).OR.
+ (SCAL(INODE,IPHASE,6)<-0.1E-12)).AND.
+ ((SCAL(INODE,IPHASE,4)>0.1E-12).OR.
+ (SCAL(INODE,IPHASE,4)<-0.1E-12)))THEN
      SCAL(INODE,IPHASE,8)=ABS(SCAL(INODE,IPHASE,5))/
+ ((SCAL(INODE,IPHASE,4)**2.+SCAL(INODE,IPHASE,6)**2.)*0.5)
      IF (((SCAL(INODE,IPHASE,6)>0.1E-12).OR.
+ (SCAL(INODE,IPHASE,6)<-0.1E-12)).AND.
+ ((SCAL(INODE,IPHASE,4)<0.1E-12).AND.
+ (SCAL(INODE,IPHASE,4)>-0.1E-12)))THEN
      SCAL(INODE,IPHASE,8)=ABS(SCAL(INODE,IPHASE,5)/
+ SCAL(INODE,IPHASE,6))
      IF (((SCAL(INODE,IPHASE,6)<0.1E-12).AND.
+ (SCAL(INODE,IPHASE,6)>-0.1E-12)).AND.
+ ((SCAL(INODE,IPHASE,4)>0.1E-12).OR.
+ (SCAL(INODE,IPHASE,4)<-0.1E-12)))THEN
      SCAL(INODE,IPHASE,8)=ABS(SCAL(INODE,IPHASE,5)/
+ SCAL(INODE,IPHASE,4))
      ELSE
      SCAL(INODE,IPHASE,8)=ABS(SCAL(INODE,IPHASE,5))
      END IF
END IF
      END IF

```

C COMPUTE CRITICAL FUNCTION FOR INCIPIENT MOTION

```

      SCAL(INODE,IPHASE,9)=COS(atan(SCAL(INODE,IPHASE,7)))
      SCAL(INODE,IPHASE,10)=1.-SCAL(INODE,IPHASE,7)/(0.577)
      SCAL(INODE,IPHASE,11)=COS(atan(SCAL(INODE,IPHASE,8)))

```

```

      SCAL(INODE,IPHASE,12)=(SCAL(INODE,IPHASE,8)**2.)/0.333333
      SCAL(INODE,IPHASE,13)=(1.-SCAL(INODE,IPHASE,12))
-   SCAL(INODE,IPHASE,14)=(ABS(SCAL(INODE,IPHASE,13)))**0.5
      SCAL(INODE,IPHASE,15)=SCAL(INODE,IPHASE,9)*SCAL(INODE,IPHASE,10)*
+   SCAL(INODE,IPHASE,11)*SCAL(INODE,IPHASE,14)
      SCAL(INODE,IPHASE,16)=((ABS(SCAL(INODE,IPHASE,15)))**0.5)*0.17

C   COMPUTE MOVABILITY NUMBER
      SCAL(INODE,IPHASE,17)=(0.09**0.25)*(TE(INODE,IPHASE)**0.5)/0.129

C   COMPUTE CONDITION I.E. DIFFERENCE
      SCAL(INODE,IPHASE,18)=(SCAL(INODE,IPHASE,17)-
+   SCAL(INODE,IPHASE,16))

C   WRITE RESULTS TO FILE CALLED 'REASON'
      OPEN(UNIT=out,FILE='REASON.DAT',STATUS='UNKNOWN',IOSTAT=ios)
c
      IF (ios/=0) THEN
        WRITE(*,*)'FILE EXISTS ALREADY - ABORT'
        STOP
      ELSE
        WRITE(UNIT=out,FMT=902)I,XP(INODE),ZP(INODE),U(INODE,IPHASE),
+   V(INODE,IPHASE),SCAL(INODE,IPHASE,17),
+   SCAL(INODE,IPHASE,18)

      END IF

200 CONTINUE

C   DEFINE FORMAT FOR IDL TO READ
902 FORMAT(I6,2X,2(F9.4,1X),2X,E10.3,2X,E10.3,2X,E10.3,2X,E10.3)

C++++++ END OF USER AREA 5 ++++++

```