# Three-Dimensional Simulation Of River Flood Flows

Volume I: Text and Tables

Hervé P. Morvan
(Eng. ESTP Paris, M.Sc.)

Thesis submitted for the degree of
Doctor of Philosophy
in the department of Civil Engineering,
The University of Glasgow

Department of Civil Engineering,
The University of Glasgow
Rankine Building
Oakfield Avenue
Glasgow G12 8LT

June 2001

# Abstract

This thesis describes the implementation of general Computational Fluid Dynamics (CFD) techniques to laboratory and natural channels under flood flow conditions. Two commercially available codes, TELEMAC and CFX4, have been used in this work.

The thesis reviews the different aspects and requirements of CFD, and proposes methods to adapt them for the simulation of complex flows found in river floods. Particular emphasis is put on: testing the level of discretization that is necessary to achieve an adequate representation of the flow; the choice and impact of the boundary conditions; convergence and scalability; and the choice of turbulence models. The conflict existing between these requirements and the complexity of the problem undertaken is also discussed.

The assessment of CFD for the calculation of flooded channel flow dynamics is carried out by simulating one laboratory test case from the Flood Channel Facility (FCF) Series B. This test case is that of a meandering two-stage channel with a depth ratio of 25% on the flood plain. Results from a computer simulation of experiment B23 are presented with a detailed quantitative comparison of the measured velocity, turbulence and bed shear stress. It supports the conclusion that CFD is able to account for the different flow mechanisms arising from the interaction between inbank and overbank flows in meandering channels. The maximum error in the prediction of the velocity is 10% and the comparisons show the calculation of bed shear stress to be reasonably accurate as well. Numerical tests indicate that the numerical solution is relatively independent of the boundary conditions, and confirm that turbulence transport is of minor importance in the experiment simulated.

Numerical results from the simulation of flood flow mechanisms in natural rivers are also presented. It is hoped that these are of value to practitioners. Two 1-km reaches on the River Severn and River Ribble are modelled. They permit the investigation of two-stage channel flow dynamics at a larger scale. The numerical verification process establishes

that the scale and the complex nature of the geometry are limiting factors, particularly for the numerical discretization of the domain and the calculation of the variables at the walls. It is however possible to estimate a priori part of the error such constraints generate. Away from the walls, the flow main features seem well predicted. The parallel between the velocity fields observed in river flood flows and those observed in the FCF is evident. Validation against field data suggests that the models are able to reproduce the flow mechanisms and account for bed shear stress variations correctly. Yet a significant level of uncertainty remains when the model predictions are compared against measured point data; more validation work is therefore required.

# Acknowledgments

I am grateful to my supervisors, Prof. G. Pender and Prof. D.A. Ervine for their guidance and constructive criticism throughout the course of this research supported by EPSRC grant No. GR/L95038. Prof. Pender was instrumental in my choice of research subject, after he introduced me to numerical modelling – nearly six years ago – enabling me to reconcile my interest in physics and applied mathematics with the endeavours of engineering.

I would also like to thank Dr. N.G. Wright, for his friendship and encouragement, and numerous electronic conversations about CFD. He arranged several visits to Nottingham University and the use of the computer facilities in his department, which undoubtedly benefited the present work.

I would like to acknowledge the advice provided by Prof. B. Younis on turbulence and mesh independence testing in the early stage of my research; Prof. R. Bettess regarding bed shear stress; and Prof. K.J. Beven, leading to rich discussions on calibration and uncertainty in distributed models. I wish to register my thanks to Mr. Kenneth McColl for his invaluable assistance with the computer facilities at the University of Glasgow.

My family, friends and colleagues gave me continuous support throughout these three years. My wife provided effortless coaching and companionship, and played a large part in enabling me to achieve my goal. Particular thanks are due to my parents for supporting my education, and my friends and colleagues at 76 Oakfield Avenue and Victoria Park Amateur Athletic Club.

Finally, this thesis is dedicated to my grandmother, Marie Morvan (1913-1998), who would have been proud to witness the completion of this work.

# Conventions

The following conventions are used consistently in the following thesis:

- Cross-sectional figures are plotted looking in the downstream direction.
- Velocity figures display the velocity magnitude or norm, calculated as the square root of the three spatial components of the velocity elevated to the power of 2.
- Angle plots refer to the direction of the velocity vector in the horizontal plane, with respect to the normal to the cross-section (which represents the 0°). Positive angles are measured when the velocity direction is heading towards the left with respect to the normal (0°), and the value of the angle is that between the velocity direction and the normal. When the velocity is heading towards the right hand side, this is considered a negative angle and measured using the same convention. These were the conventions used by the FCF experimentalists.
- All units are SI units unless stated otherwise.

All the figures are presented in Volume II of the present thesis. The tables are inserted in the text at the exception of Table 3.1 placed at the end of Chapter 3 for practical reasons.

# Table of Content

# CHAPTER 6

# Application of CFD to Flooded Rivers – Rivers Severn and Ribble

## CHAPTER 7

## Conclusions and Recommendations

# List of Tables

The tables are included in the text, at the exception of Table 3.1 placed at the end of Chapter 3.

# List of Figures

The figures are placed in the following order in Volume II of the present thesis.

# Chapter 1: Introduction

# Chapter 1: Introduction

*"Floods account for about a third of all natural catastrophes. They cause more than half of all the fatalities".*

(Reinsurance Munich Company, 1998 and 1997, in Whyte, 1999)

*"From a geomorphological viewpoint, flows in channels and over land surfaces are one of the dominant factors in producing change. If these changes are to be quantified then some knowledge of hydraulic engineering is required".*

(Knight, 1989)

## 1.1 FLOODING: PROBLEM STATEMENT

The main factor affecting flooding is the humane factor.

Farming and industry require the proximity of water sources, and populations have consequently concentrated around them. As a result, more than half of the world population lives within 60 km of the coast, with most major cities either located on the coast or on a major river system. The proximity of human settlements and the pressure on land use nearby rivers have lead to a conflicting situation where the danger of flooding has become increasingly unacceptable to the population. This proximity has reflected in the need to understand and control the river flows, in order to protect the population and understand the effects of high flows on sediment and pollutant transport, on the stability of the river system and on the reliability of defence measures.

In a first step, flood control has lead to the channelisation of the river systems and the construction of large embankments and dams in a more or less empirical fashion. In Britain alone, following industrialisation, it has been estimated that about 60% of the lowland rivers have been modified (Brookes, 1988). These modifications have had a profound impact on the river hydrology and geomorphology, and have affected their ability to convey flood flows. Simple methods to relate water discharge to water levels have initially been developed, mostly based on a one-dimensional analysis of the flood event along the channel.

More recently our approach to rivers has changed and technology allows a better insight into complex fluid flow mechanics. It is now acknowledged that natural channels are instrumental to the sustainability of a healthy and diverse environment. A lot of emphasis is put on river conservation and restoration. The priorities have shifted from that of an "industrial" control of the rivers to that of a better, less empirical understanding of natural channels. This has lead researchers to investigate two- and three-dimensional dynamics of two-stage flows in order to incorporate a more complete description of the flow to design better structures and defence systems, and alleviate the effects of the flood wave transit. At first, scale experiments have been used, e.g. Toebes and Sooky (1967) or the Flood Channel Facility programme (SERC, 1993). In recent years, Computational Fluid Dynamics (CFD) has successfully reproduced some of these experiments, notably for inbank channels and straight two-stage channels (Cokljat, 1993; Wright and Morvan 2000), paving the way for a more ambitious undertaking, that of modelling a full river reach three-dimensionally. The use of CFD to resolve inbank flow condition problems in natural rivers has recently been investigated (Sinha *et al.*, 1998 , Bradbrook *et al.*, 1998). Yet no contribution has been made, so far, regarding the application of such a technique to complex three-dimensional flood flows, despite the fact that they are still poorly understood and highly disruptive.

## 1.2 TWO-STAGE CHANNEL HYDRODYNAMICS

### 1.2.1 Experimental Results

Most of the current knowledge regarding flood hydrodynamics relies on the outcome of the Flood Channel Facility (FCF) research programme. Key papers by Knight and co-workers (1989, 1996), Willets and co-workers (1990, 1993) and Ervine *et al.* (1993) have presented these results for meandering two-stage channels. This work has been pursued by the group of Shiono and Muto (1998a, 1998b, 1999) using more modern measurement apparatus to deliver an even clearer understanding of the three-dimensional flow structures and make the link with numerical modelling.

What has been the focus of most of these research programmes is the distribution and transfer of energy between inbank and floodplain flow, as well as the development of flow structures particular to flood flows. It has emerged that the hydrodynamics of flooded channels is quite different from their inbank counterparts. In particular interface vortices and large-scale vortex structures develop at the banks for straight channels. Longitudinal vorticity, turbulence and subsequent secondary flow features have also been found to be altered by floodplain flows and vary with the floodplain water depth (Knight and Shiono, 1996). For the case of meandering channels, of interest here, the interaction between floodplain and inbank flows has strong implications as the overbank flow tends to travel in a skewed direction with the main channel. This has been found to result in a reversal of the secondary flow circulation at the bend apex due to the vorticity generated in the main channel by the overbank flow crossing over the main channel at an angle upstream of the bend (Ervine *et al.*, 1993). As a result of the floodplain-inbank flow interaction the direction of travel of the flow on the floodplain does not follow the steepest slope line, but gets diverted by the inbank flow momentum and the water that is ejected from the main channel. In places at the cross-over water is plunging vertically in the main channel from the floodplain. This is revealing of the three-dimensional nature of the flow. These flow structures have a profound effect on the shear stress distribution at the boundaries (Knight and Shiono, 1996). They must affect sediment and pollutant transport in the river, have some impact on the river morphology, and also play a role in the overall flood dynamics.

More details about the FCF results are given in Chapter 5.

### 1.2.2 Two-Stage Channels and Flood Flow Modelling

Simple two-stage channel experiments have enabled engineers to gain a significant insight into the three-dimensional mechanics of such flow. It is believed that such knowledge could help determine which transport mechanisms are predominant in different flood flow situations (turbulence, advection). It could also be applied to natural channels where a suitable three-dimensional representation of the flood flow structures could prove essential to the design of effective defences and structures, river bank stability and the preservation of the ecosystem local habitat.

At the same time the knowledge gained from the FCF laboratory observations also needs to be verified to determine whether flow structures similar to that in the FCF would occur in natural channels. Since a complete experimental investigation of the flow hydrodynamics in a flooded meandering river would be painstaking, a numerical analysis using state-of-the-art three-dimensional Computational Fluid Dynamics (CFD) techniques, supported by validation field data, is suggested here. Such an investigation would position itself in the line of the FCF programme with the double objective to apply CFD to flooded open-channels and verify whether a connection exists between the FCF and large-scale flood flow. Details about the proposed investigation are given in Section 1.3. The code governing equations are presented in Chapter 2.

## 1.3 SCOPE OF THE PROJECT AND OUTLINE OF THE THESIS

The present study is a three-dimensional computational investigation of the spatial flow mechanisms and structures developing in meandering and natural two-stage channels during flood events. Laboratory and field data are used to validate the numerical models and demonstrate the accuracy of the results as well as the potential of CFD in river engineering.

The principal objectives of the research can be listed as:

(i) the application of state-of-the-art Computational Fluid Dynamics (CFD) techniques to a new environmental flow problem, flooding, and their evaluation against experimental and natural channel data including detailed velocity, turbulence and bed shear stress measurements;

(ii) the validation (or improvement) of the current knowledge regarding two-stage channel hydrodynamics in rivers using CFD experimentation at small and large scales;

(iii) the presentation of a first guide regarding the modelling of river flows in three-dimensions, of interest to practitioners.

In order to achieve these objectives and address the question of the applicability of CFD to practical engineering problems it was decided to make use of two validated commercial codes: TELEMAC (v. 2.2) and CFX (v. 4.2 and 4.3). They provided the technological toolbox used to move computational river engineering forward. TELEMAC was chosen because of its fluvial hydraulics background and in-built ability to calculate the free-surface variations. CFX, on the other hand, is a general CFD code. It was preferred over other codes such as FLOW3D or FLUENT because of: the range of models and algorithms included in the code; its ability to deal with roughness; and the existence of a range of hydraulic applications, due, among other things, to collaborative work with HR Wallingford (Lavedrine, 1996). Both codes will be fully described in Chapter 4.

This thesis is structured as follows:

Chapter 2 introduces the reader to the mathematical equations and models used to describe the flow.

Chapter 3 is a review of previous attempts to model open-channel flows numerically. The underlying assumptions are presented, and their relevance to the present work is highlighted.

All the important numerical considerations regarding the computational analysis of the fluid flow are reviewed in Chapter 4. This chapter offers comprehensive detail on all the important aspects of model construction, numerical discretisation, definition of the boundary conditions, and solution strategy applied in this thesis.

Chapters 5 and 6 present the results obtained by the author during the course of his PhD. Chapter 5 details the result obtained while reproducing the FCF experiment B23 numerically. It is presented as an evaluation test to assess the capacity of the codes to reproduce two-stage channel flows accurately. Detailed velocity and turbulence laboratory measurements are used to judge the quality of the model, and new explanations are put forward regarding the nature of turbulence in such flow. This work has been disseminated in the form of two conference papers (Morvan *et al.*, 2000a, 2000b) and a journal paper to appear soon (Morvan *et al.*, 2000c)

In Chapter 6 CFD models for two 1-km reaches on the River Severn and River Ribble are presented under flood flow conditions. This chapter is used to check whether the flow structures observed during the FCF experiments are repeated at larger scales, and assess how practical it is to use CFD for environmental river flow problems. Some validation is conducted in the case of the River Severn giving a more accurate idea of the model capacity. The results of the River Severn study case will be presented at a conference later this year (Morvan *et al.*, 2001).

The overall conclusions are summarised in Chapter 7, together with some ideas for future work to be undertaken.

## 1.4 REFERENCES FOR CHAPTER 1

1. Bradbrook, K.F., Biron, P.M., Lane, S.N., Richards, K.S., Roy, A.G., (1998), "Investigation of Controls on Secondary Circulation in a Simple Confluence Geometry Using a Three-Dimensional Model", Hydr. Processes, Vol. 12, pp. 1371-1396.
2. Brookes, A., (1988), "Channelized Rivers: Perspectives for Environmental Management", Chichester : Wiley.
3. Cokjlat, D., (1993), "Turbulence Models for Non-Circular Ducts and Channels", PhD Thesis, City University, London.
4. Ervine, D.A., Willetts, B.B., Sellin, R.H.,J., Lorena, M., (1993), "Factors Affecting Conveyance in Meandering Compound Flows", J. Hydr. Eng., Vol. 119, pp.1383-1398.
5. Knight, D.W., Shiono, K., (1996), "River Channel and Floodplain Hydraulics", in "Floodplain Processes", M.G. Anderson, D.E. Walling, P.D. Bates (Eds.), John Wiley and Sons Ltd., pp. 139-181.
6. Knight, D.W., (1989), "Hydraulics of Flood Channels", in "Floods: Hydrological, Sedimentological and Geomorphological Implications", K. Beven and P. Carling (Eds.), John Wiley and Sons Ltd.
7. Lavedrine, I., (1996), "Evaluation of 3D Models for River Flood Applications", Report TR6, HR Wallingford.
8. Morvan, H., Pender, G., Ervine, D.A., (2001), "Three-Dimensional Simulation of River flood Flows", Proc. of the First Int. Conf. On River Basin Management, Cardiff, September 2001 (accepted for publication).
9. Morvan, H., Wright, N.G., Pender, G., Ervine, D.A., (2000a), "Three-Dimensional Modelling of Secondary Currents in Meandering Open-Channels", CFX Academic Conference, Theale, March 2000, Publication available online www.aeat.com/cfx.
10. Morvan, H., Pender, G., Wright, N.G., Ervine, D.A., (2000b), "Three-Dimensional Modelling of the Flow Mechanisms in Flooded Meandering Channels", Proceedings of the Int. Symp. On Flood Defence, Kassel, September 2000, Herkules Verlag, Kassel, Vol. 1, pp. D153-D161.

11. Morvan, H., Pender, G., Wright, N.G., Ervine, D.A., (2000c), "Three-Dimensional Hydrodynamics of Meandering Compound Channels" (submitted to ASCE).
12. Muto, Y., Shiono, K., Imamoto, H., Ishigaki, T., (1998), "Three-Dimensional Flow Structure for Overbank flow in meandering Channels", J. of Hydroscience and Hydr. Eng., Vol. 16, No. 1, pp. 97-108.
13. SERC, (1993), SERC FCF Series B Report, Vol. 1 to 8.
14. Shiono, K., Muto, Y., Knight, D.W., Hyde, A.F.L., (1999), "Energy Losses due to secondary Flow and Turbulence in Meandering Channels with Overbank Flows", J. Hydr. Res., Vol. 37, No. 5, pp. 641-664.
15. Shiono, K., Muto, Y., (1998), "Complex Flow Mechanisms in Compound Meandering Channels with Oberbank Flow", J. Fluid Mech., Vol. 376, pp. 221-261.
16. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.
17. Toebes, G.H., Sooky, A.A., (1967), "The Hydraulics of Meandering Rivers with Floodplain", Proc. A.S.C.E. J. of Waterways and Harbours, Vol. 73, pp. 213-236.
18. Willetts, B.B., Hardwick, R.I., (1993), "Stage Dependency for Overbank Flow in Meandering Channels", proc. of the Inst. Civ. Eng., Water, Marit. and Energy, Vol. 101, No. 1, pp. 45-54.
19. Willetts, B.B., Hardwick, R.I., (1990), "Model Studies of Overbank Flow from a Meandering Channel", Proc. Int. Conf. on River Flood hydraulics, W.R. White (Ed.), John Wiley and Sons Ltd., pp. 173-183.
20. Wright, N.G., Morvan, H. (2000), "CFD Model of the Riprap Test Facility", (to appear in ASCE monograph edited by S. Wang)

# Chapter 2:
# Governing Equations and Turbulence Models

# Chapter 2:

# Governing Equations and Turbulence Models

*"An ideal model should introduce the minimum amount of complexity while capturing the essence of the relevant physics".*

(Wilcox, 1993)

## 2.1 GOVERNING EQUATIONS: RANS EQUATIONS

The water fluid is considered as Newtonian, incompressible and taken to have constant physical properties. It is governed in the present investigation by a time-averaged formulation of the Reynolds-Averaged Navier-Stokes (RANS) equations, as follows:

$$\nabla \cdot \left(\rho \vec{U}\right) = 0 \qquad (2.1)$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot \left(\rho \vec{U} \otimes \vec{U}\right) = \vec{B} + \nabla \cdot \left(\sigma - \rho \overline{\vec{u} \otimes \vec{u}}\right) \qquad (2.2)$$

With,

$$\sigma = \mu\left(\left(\nabla \vec{U}\right) + \left(\nabla \vec{U}\right)^{T}\right) - p\delta_{ij} \qquad (2.3)$$

(CFX, 1997)

Where, $\rho$ = fluid density; $\mu$ = the molecular viscosity; $\vec{U} = (U_i)_{i=(x,y,z)}$ the time-averaged velocity field; $\vec{B}$ = body force; $\sigma$ = stress tensor matrix; $\rho \overline{\vec{u} \otimes \vec{u}}$ = the so-called Reynolds stresses, $p$ = pressure and $\delta_{ij}$, the Kronecker delta symbol.

The time-averaged formulation approximates flow turbulence through the introduction of a suitable turbulence model and constitutes the most practical approach for the simulation of three-dimensional flow problems in engineering. Turbulence closure models are needed to formulate the Reynolds stresses in terms of known or measurable quantities. The turbulence modelling method used in the current work is discussed in section 2.2.

More rigorous methods exist such as the Direct Numerical Simulation (DNS) of the Navier-Stokes equations, which is an exact approach. Alternatively, Large Eddy Simulation (LES) makes use of the physics of turbulence to simplify the DNS approach by using a hybrid of DNS to solve for large scale turbulent structures, and a spatially-averaged formulation of the Navier-Stokes equations to model the smaller isotropic components of turbulence (Ferziger and Peric, 1996). These methods will be presented for completeness in section 2.3 together with other approaches such as chaos. They are not be used in the course of the present work.

## 2.2 TURBULENCE CLOSURE MODELS FOR THE RANS EQUATIONS

### 2.3.1 Boussinesq Relationship

Since the time-averaged RANS equations are used a closure model is necessary to provide either a transport equation or an algebraic expression for the Reynolds stresses. In the present work, Boussinesq eddy viscosity hypothesis is used for the zero- and two-equation turbulence models, to yield an expression analogous to (2.3):

$$\rho\overline{\vec{u}\otimes\vec{u}} = \rho\nu_t\left((\nabla\vec{U})+(\nabla\vec{U})^T\right)+\frac{2}{3}\cdot\left(-\rho k-\rho\nu_t\nabla\cdot\vec{U}\right)\delta_{ij} \tag{2.4}$$

Where $k = 0.5\cdot|\vec{u}|^2$ is the turbulent kinetic energy and $\nu_t$ is the turbulent eddy viscosity.

Equation (2.4) assumes a direct relationship between Reynolds stresses and the mean strain rate, although such alignment is not physical (Lumley, 1994). Models of the type $k$-$\varepsilon$ are calibrated to give the correct value to the off-diagonal stress terms in (2.4). This implies that the normal stresses must be wrong, which is not important in simple turbulence events. However, in flows where the normal stresses in (2.4) do matter, such relationship is bound to fail and it is not clear as to how to calibrate such models for more complex situations such as flow separation for example (Lumley, 1994). Therefore one has to accept and be aware of model (2.4)'s limitation.

### 2.2.2 Isotropic Turbulence: The Mixing-Length Model

In its simplest expression, turbulence can be characterised by two parameters, usually taken as a velocity scale (U) and a length scale ( $\ell$ ). Therefore one can assume:

$$\nu_t = CU\ell \tag{2.5}$$

Where, $C$ is proportionality constant.

Large eddies in the flow contain most of the turbulent kinetic energy. It is therefore reasonable to view the turbulence length scale as being related to the large eddy dimensions, and to assume that if they interact with the mean flow then the characteristic velocity scale should be chosen as a function of the eddy's properties and the mean flow properties. Since large eddies are enhanced by and interact with mean flow principally through its velocity gradient, then:

$$\nu_t = C \cdot \left( c\ell \left| \frac{\partial \bar{U}}{\partial z} \right| \right) \cdot \ell = \chi \cdot \ell_m^{\ 2} \cdot \left| \frac{\partial \bar{U}}{\partial z} \right| \tag{2.6}$$

Where c is another constant; $\chi$ = is a "damping" function varying between 1 and 0 (given in Janin *et al.*, 1997, Fig. 2, p.19), which is often taken equal to 1 when the fluid has constant physical properties; $\ell_m$ = the characteristic length scale is called the mixing-length.

The above model is obviously a simplification in that it will only account for mean flow properties and will not model recirculating velocities. Its simplicity however, makes it easy to implement especially on large domains such as coastal areas and long river system stretches. Because turbulence is a function of the flow, it is necessary to adjust the model for different situations by varying the mixing length value. This is a difficult task. Alfrink (1982) and Sauvaget and Usseglio-Polatera (1987) recommended:

$$\ell_m = 0.2 \cdot \kappa h \qquad (2.7)$$

Where $\kappa$ is von Karman constant and $h$ the water depth. In TELEMAC, finite element code discussed later Janin *et al.* used (2.7) together with the condition at the bed:

$$\ell_m = \kappa \cdot \Delta h \ \textbf{if} \ \Delta h < 0.2 \cdot h \qquad (2.8)$$

Where $\Delta h$ is the distance from the bottom boundary. In TELEMAC equation (2.6) is only used for the vertical diffusion, the horizontal values remaining constant and the effective viscosity, $\nu_e = \mu/\rho + \nu_t$, chosen equal to $10^{-2}$ to $10^{-4}$.

A full discussion of the issues and other alternative formulations can be found in Rodi (1980).

### 2.2.3 Isotropic Turbulence: The Standard $k$-$\varepsilon$ Model

Turbulence is not as well-behaved as equations (2.6)-(2.8) would suggest. It is a dynamic process, and should therefore be modelled as such, i.e. without prior knowledge of the turbulence structure. A way forward is therefore to consider the dynamic transport of two basic turbulence quantities and use these to calculate the mixing length or eddy viscosity. The first is a velocity scale (or the turbulence kinetic energy "$k$") and the second, the dissipation of turbulence kinetic energy, "$\varepsilon$" (Chou, 1945). In this case the eddy viscosity calculated using equation (2.9) is a spatially varying scalar, which assumes an isotropic representation of turbulence via equation (2.4):

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \qquad (2.9)$$

The standard turbulent kinetic energy transport equation is obtained by multiplication of the instantaneous Navier-Stokes equations by each fluctuating velocity component, to obtain three equations, which are then added. This is followed by a repeat of this process on the time averaged Navier-Stokes equations, subtraction of the two resulting sets of equations and re-arrangement to yield the equation for $k$ (Tennekes and Lumley, 1972). A similar approach is used to derive a transport equation for the dissipation $\varepsilon$. However, this derivation is far more empirical and the simplifications required to model its terms so severe (referred to as "*drastic surgery*", Wilcox, 1993, p.89) that it is best to consider the entire resulting equation as a model.

$$\overbrace{\frac{\partial \Phi}{\partial t}+\nabla\cdot\left(\vec{U}\Phi\right)}^{\text{Advection}}-\overbrace{\nabla\cdot\left(\left(\frac{\nu_t}{\sigma_\Phi}\right)\nabla\Phi\right)}^{\text{Diffusion}}=\overbrace{S_\Phi}^{\text{Source}} \tag{2.10}$$

With,

| $\Phi$ | $\sigma_\Phi$ | $S_\Phi$ |
|---|---|---|
| $k$ | $\sigma_k$ | $P_k - \varepsilon$ |
| $\varepsilon$ | $\sigma_\varepsilon$ | $\frac{\varepsilon}{k}\left(C_{1\varepsilon}P_k - C_{2\varepsilon}\varepsilon\right)$ |

**TABLE 2.1 – Standard *k-ε* Transport Equation Terms as in Equation (2.10)**

And,

| $C_\mu$ | $\sigma_k$ | $\sigma_\varepsilon$ | $C_{1\varepsilon}$ | $C_{2\varepsilon}$ |
|---|---|---|---|---|
| 0.09 | 1.0 | 1.30 | 1.44 | 1.92 |

**TABLE 2.2 - Standard *k-ε* Turbulence Model Constants**

$$P_k = -\overline{\vec{u}\otimes\vec{u}}\cdot\nabla\vec{U} \tag{2.11}$$

Where $P_k$ = the shear production term.

Discussion

There exists other versions of the $k$-$\varepsilon$ model, and indeed other two-equation turbulence models, in particular for low or high Reynolds numbers. For more details on these models (e.g. the RNG $k$-$\varepsilon$ model or the $k$-$\omega$ model) the reader is referred to Wilcox (1993) and CFX (1997).

Of all the turbulence models, the standard $k$-$\varepsilon$ is by far the most widely used in industry. Its low computational cost and relative numerical stability have made it very popular, despite its main flaws arising from the assumption of isotropic turbulence and the reservations regarding the adequacy of the dissipation equation. It has been particularly popular in river modelling, where to date the three-dimensionality and the impact of turbulence have largely been ignored. In fact, there are recent successful examples of three-dimensional simulations of semi-natural and natural channels using the standard $k$-$\varepsilon$ model (Demuren, 1993; Sinha *et al.*, 1998).

To date more complex models have only been applied to free-surface inbank flows in prismatic channels. In some of these attempts the need to account for anisotropy-driven flow features has justified the implementation of higher order turbulence models such as non-linear $k$-$\varepsilon$ models (Manson, 1994; Stansby and Zhou, 1998) and Reynolds stress models (Cokljat, 1993; Cokljat and Younis, 1995).

### 2.2.4 Anisotropic Turbulence: The Reynolds Stress Model (RSM)

Anisotropic turbulence has been observed in experimental measurements and DNS. These observations suggest that the relationship between the Reynolds stresses and strain rate is not as simple as implied by equations (2.9) and (2.4). To overcome this it is necessary to abandon the eddy-viscosity approach of equation (2.4) to improve the description of the Reynolds stresses, and formulate a more complete model for each of them.

The most complex model in use today is the Reynolds Stress Model (RSM), which solves one transport equation for each of the six Reynolds stresses and therefore accounts for turbulence anisotropic features. It is obtained after multiplication of the Navier-Stokes equations for the instantaneous velocity components by the transpose of the Reynolds stresses. These equations are then summed and time-averaged to yield:

$$\overbrace{\frac{\partial \overline{\vec{u}\otimes\vec{u}}}{\partial t}+\nabla\cdot\left(\overline{\vec{u}\otimes\vec{u}}\otimes\vec{U}\right)}^{\text{advection}}=-\overbrace{\left(\overline{\vec{u}\otimes\vec{u}}\left(\nabla\vec{U}\right)+\overline{\vec{u}\otimes\vec{u}}\left(\nabla\vec{U}\right)^{T}\right)}^{\text{production}=\mathrm{P_s}}$$

$$\overbrace{-\nabla\cdot\left(\overline{\vec{u}\otimes\vec{u}\otimes\vec{u}}+\frac{1}{\rho}\left(\overline{p'\vec{u}I+p'\vec{u}^{T}I}\right)-\nu\nabla\cdot\left(\overline{\vec{u}\otimes\vec{u}}\right)\right)}^{\text{turbulence transport diffusion}} \quad (2.12)$$

$$\underbrace{-2\nu\left(\overline{(\nabla\vec{u})\otimes(\nabla\vec{u})}\right)}_{\text{dissipation}}+\underbrace{\overline{\frac{p'}{\rho}\left(\left(\nabla\vec{u}\right)+\left(\nabla\vec{u}\right)^{T}\right)}}_{\substack{\text{pressure - strain}\\ \text{redistribution} = \Phi_{ij}}}$$

Where $P_s$ is the shear production tensor, $p'$ is the pressure fluctuation, and $\Phi_{ij}$ is the pressure strain correlation term. In equation (2.12) only the advection, production and viscous diffusion terms can be treated in their exact forms.

Diffusion

In the diffusion terms, the triple-velocity correlation and the pressure fluctuation terms need to be modelled and expressed as functions of known or knowable quantities such as the mean velocity field ($\vec{U}$), the Reynolds stresses ($\overline{\vec{u}\otimes\vec{u}}$) or the dissipation of turbulent kinetic energy ($\varepsilon$). Following evidence from Hanjalic and Launder (1972a), the pressure fluctuation term in the turbulence diffusion transport is neglected in the following.

Darly and Harlow (1970) proposed a formulation based on the gradient transport hypothesis for the triple-velocity correlation (referred to as DH formulation hereafter):

$$\overline{\vec{u}\otimes\vec{u}\otimes\vec{u}}=C_s\frac{k}{\varepsilon}\overline{\vec{u}\otimes\vec{u}}\left(\nabla\overline{\vec{u}\otimes\vec{u}}\right) \quad (2.13)$$

Hanjalic and Launder (1972b) derived a more general formulation that can be expressed as equation (2.14) after simplification and numerical approximations (referred to as HL formulation):

$$\overline{\vec{u}\otimes\vec{u}\otimes\vec{u}} = C_s \frac{k}{\varepsilon}\overline{\vec{u}\otimes\vec{u}}\left(\nabla\overline{\vec{u}\otimes\vec{u}}\right)^{T} \tag{2.14}$$

Cokljat (1993) preferred the DH formulation based on evidence by Launder *et al.* (1975), Samaraweera (1978) and Gibson and Younis (1982) who found that, in general, it produced better results. Schwarz and Bradshaw (1994) however, point out that it is difficult to argue that the DH formulation is "*the best overall solution for 3-D flow*" as it appears that the transport gradient concept behind (2.13) and (2.14) is inadequate (Younis *et al.*, 1999). Yet, in general, most formulations found in CFD for the triple-velocity correlation rely on it (see Schwarz and Bradshaw for a review, 1994) and equation (2.14) is implemented in the present work.

Dissipation

There is greater consensus as to how dissipation should be modelled. Rotta's model (1951) is widely accepted. As dissipation occurs at small length scales it is isotropic and consequently:

$$2\nu\left(\overline{(\nabla\vec{u})\otimes(\nabla\vec{u})}\right) = \frac{2}{3}\varepsilon I \tag{2.15}$$

The dissipation of turbulent kinetic energy is obtained from its own transport equation in a similar form to (2.5). Following Hanjalic and Launder (1972b), the diffusion term in equation (2.9) is modified to yield (see also Table 2):

$$\frac{\partial\Phi}{\partial t} + \nabla\cdot\left(\vec{U}\Phi\right) - \overbrace{\nabla\cdot\left(C_\varepsilon \frac{k}{\varepsilon}\overline{\vec{u}\otimes\vec{u}}\cdot(\nabla\Phi)\right)}^{\text{Diffusion}} = S_\Phi \tag{2.16}$$

Pressure-Strain Correlation

The pressure-strain expression is difficult to model. It has been calculated by Chou (1945) and has three components (Launder *et al.*, 1975): one that represents purely turbulent interactions ($\Phi_{ij,1}$, e.g. eddies interaction), one that accounts for the interaction between the mean velocity and the fluctuating velocities strains ($\Phi_{ij,2}$, e.g. eddies and the mean flow), and a third that represents the effects of the proximity of the wall on the pressure-strain term ($\Phi_{ij,w}$):

$$\overline{\frac{p'}{\rho}\left(\left(\nabla\vec{u}\right)+\left(\nabla\vec{u}\right)^{T}\right)}=\Phi_{ij,1}+\Phi_{ij,2}+\Phi_{ij,w} \tag{2.17}$$

The first component also called the "slow" part has been modelled by Rotta (1951) as:

$$\Phi_{ij,1}=-C_1\frac{\varepsilon}{k}\left(\overline{\vec{u}\otimes\vec{u}}-\frac{2}{3}kI\right) \tag{2.18}$$

The second component, or "rapid" part, follows the work of Launder *et al.* (1975):

$$\Phi_{ij,2}=-C_2\left(P_s-\frac{2}{3}P_k\right) \tag{2.19}$$

This version in (2.19) however, is a simplification using only the first part of the model equation presented by Launder *et al.* (1975). As reported in Cokljat and Younis (1995), equation (2.19) has been successfully used in several applications. These authors resorted to the use of the full model equation to simulate straight channel flows "*in what must be one of the very few flows in which this was found to be necessary*". This was due to the fact that they were trying to account for small size secondary recirculations in straight channels that are mainly generated by turbulence anisotropy.

The wall reflection term $\Phi_{ij,w}$ as it is traditionally modelled (Launder *et al.*, 1975; Naot and Rodi, 1982) has been reported as the "*main obstacle to the use of the complete Reynolds-stress-transport in practical engineering*" (Basara and Cokljat, 1995). This comes from the fact that the normal to the wall needs to be calculated at all points, which is both expensive and a source of hysteresis where the normal to the wall is discontinuous. Several authors have consequently adopted models for the pressure-strain correlation without a wall-reflection term (Speziale *et al.*, 1991; Launder and Li, 1994). Launder and Li (1994) used experimental results from DNS showing that the "slow" process was much less influenced by pressure reflection than the "rapid" component. Consequently their strategy has been to incorporate additional terms and remodel $\Phi_{ij,2}$. In their application case this lead to a simple expression similar to (2.19)

$$\Phi_{ij,2} = -0.6\times\left(P_s - \frac{2}{3}P_k\right) \tag{2.20}$$

Their square duct flow model using (2.19) yielded satisfactory results compared with numerical work making use of wall reflection terms.

Summary

The RSM model that is implemented in the current thesis is composed of the following sub-models:

(i) The Hanjalic and Launder (HL, 1972b) formulation for the triple-velocity-correlation (2.14);

(ii) Rotta's model (1951) for the dissipation (2.15), together with the transport equation modified by Hanjalic and Launder (1972b) for the dissipation of the turbulent kinetic energy (2.16);

(iii) Launder, Reece and Rodi's model (LRR, 1975) for the pressure strain correlation without wall-reflection terms (2.20) (Launder and Li, 1994).

It mostly relies on the original model presented by Launder *et al.* (1975), and uses the following constant values:

| $C_1$ | $C_2$ | $C_s$ | $C_{1\varepsilon}$ | $C_{2\varepsilon}$ |
|---|---|---|---|---|
| 1.8 | 0.6 | 0.22 | 1.44 | 1.92 |

**TABLE 2.3 – RSM Constants**

Discussion

The RSM has a far greater universality than models based on the eddy viscosity concept. This is due to its more rigorous and detailed mathematical formulation. It is also easier to incorporate a better description of the physics of the flow in a greater number of equations. The fact that it is the most complete model available made it an interesting reference to investigate the influence of turbulence on the solution in the present study. Yet this model still has many inadequacies:

(i) it uses the same dissipation ($\varepsilon$) equation as the $k$-$\varepsilon$ model, with its intrinsic flaws (Wilcox, 1993);

(ii) the closure form of the pressure-strain tensor (in particular the wall-reflection terms) is not satisfactory and raises a doubt over the applicability of the model;

(iii) the partial differential equations are often very stiff (Wilcox, 1993), which impairs numerical stability and convergence, especially in complex natural flows.

The first point is a major flaw in the sense that a sound mathematical formulation of the Reynolds stresses is coupled to a coarsely modelled equation. In fact Wilcox (1993, p.89) implies that the dissipation transport equation should be viewed rather as "*an empirical equation for the rate of energy transfer from the large eddies (equal, of course, to the rate of dissipation of the small eddies)*". This is linked to two interrelated problems: the impossibility of measuring some of the most complex terms in the exact dissipation equation using conventional techniques (Wilcox, 1993); and the resulting lack of understanding regarding the energy spectrum for the dissipative process. Direct Numerical Simulation (DNS) should help shed some light on these terms (Mansour *et al.*, 1988).

Secondly, if there is a general broad agreement on the modelling of the turbulent transport diffusion and diffusion terms in equation (2.12), the wall-reflection term is more open to uncertainty. At the walls, the isotropisation process of the small scale eddies, correctly predicted in non-wall bounded flows, is damped to allow the turbulence to become increasingly anisotropic due to the presence of the walls. This problem is further compounded by the use of an isotropic dissipation model. Wilcox (1993) reports that there is no clear understanding as to why this term should have an impact throughout a channel flow. It is argued that the echo effect could scale with maximum eddy size, which, for channel flow, would be of the order of the channel size. In addition, some of the test results would allow inference that the pressure-strain shortcomings could in fact reflect shortcomings of the dissipation equation near the wall. It is therefore very difficult to establish the relevance of such a term in the current open-channel problem. At present the only known alternative to the LRR model is to expand it using non-linear pressure-strain terms that asymptotically follow the correct distribution of stresses as the wall is approached. An example is the model of Speziale, Sarkar and Gatski (SSG, 1991). Unfortunately, as reported by Speziale (1994), the use of such a non-linear model is not without problems either, even for simple flows.

The third issue is particularly important with regard to the practical implementation of such a technique. As the mean flow time scale differs from the turbulence time scale by several orders of magnitudes, the equations for both the $k$-$\varepsilon$ and RSM are much stiffer than for laminar flow. An equation is said to be numerically stiff when there are two or more very different scales of the independent variable on which the dependent variables are changing (Wilcox, 1993). This problem has to be tackled using an uncoupled solution of the momentum and pressure correction, and turbulence equations. Adequate time steps (or under-relaxation factors for steady state solutions) have to be chosen.

### 2.2.5 Anisotropic Turbulence: Some Alternatives to the RSMs?

An alternative to formulating complete transport equations such as in the RSMs is to use extended non-linear relationships of the Boussinesq assumption presented in (2.4), so that the Reynolds stresses are not directly proportional to the mean strain rate. This constitutes a compromise between the simplicity offered by an algebraic relation such as (2.4) and the need for anisotropy in the model. Such relationships lead to models where the anisotropy of the normal stresses is indeed taken into account in the non-linear extension. They should theoretically have a far greater applicability than the standard two-equation model presented in section 2.2.3. Unfortunately this is not the case.

Manson (1994) presents a review of the most recent non-linear $k$-$\varepsilon$ models, which he tested in the case of open-channels flows. Although such models are only 20 to 30% more costly than the standard $k$-$\varepsilon$ model and require less storage space than RSMs, he points out that their usefulness and applicability are not fully known. It seems indeed that they remain problem dependent and suffer from stability problems (the models are very stiff). Problem dependency seems *a posteriori* quite evident since the author understands several of the non-linear extensions appear to have been formulated with a specific problem improvement in mind. In addition, non-linear model deficiency has a lot to do with the constitutive equation (2.4) basic deficiency, as pointed out by Wilcox (1993). The results of Lin and Shiono (1995) show that a non-linear $k$-$\varepsilon$ gives a better prediction of the normal isovel than the standard linear version at the bank interface of a straight two-stage channel. However, when detailed comparisons are made for the lateral velocity, bed shear stress, turbulent eddy viscosity, and tracer concentrations, it is often difficult to pinpoint significant differences. These shortcomings, and in particular the lack of "universality" of the models, suggest to the author that more research is required to determine a suitable model for open-channel flow.

Another more rigorous form of non-linear relationship for the Reynolds stresses can be obtained by reduction of a RSM. Although it provides satisfactory results for flows over curved surfaces, such models (called Algebraic Stress Model, ASM) fail to properly account for sudden changes in the mean strain rate, a shortcoming that is common to most models based on the Boussinesq assumption. Wilcox (1993) provides a brief review concerning the formulation of ASMs illustrated by several applications.

Because the RSM formulation is more widely acknowledged, and that the extra computational effort required to run it appears to be the same as for the non-linear models (see Chapter 5), it was judged more appropriate to use a RSM in the present work.

## 2.3 OTHER FORMULATIONS OF THE TURBULENT FLOW PROBLEM

This section presents ways of simulating turbulent flows by means other than using the RANS combined with a turbulence closure model. These methods remain in the domain of research and have had little application in practical engineering so far. They have not been used in the forthcoming research work because they do not meet the engineering requirements of "applicability to natural channels" that are central to the author's objectives. Nevertheless with the development of computer technology and mathematical science, the following approaches might become more frequently used to calculate and understand turbulent flows in the near future.

### 2.3.1 Direct Numerical Simulation (DNS)

This approach is the easiest to define from a conceptual point of view, but it is the most computationally expensive and memory demanding. It is the most exact approach as it solves the Navier-Stokes equations without averaging or approximation other than the necessary numerical discretizations whose errors can be estimated and controlled (Ferziger and Peric, 1996).

In such simulation all flow motions, down to the Kolmogorov microscale $\eta$, are resolved in a complete time-dependent solution of the Navier-Stokes equations. Hence the domain must be at least as large as the largest turbulent eddy, which measure is taken to be the so-called integral scale, $L$, in statistical turbulence theory. Assuming homogeneous turbulence, the domain resolution has to be equal to $L/\eta$ at a minimum, in each direction, i.e. $(L/\eta)^3$ in three dimensions. It can be shown (Wilcox, 1993) that this ratio is approximately equal to $(3 \cdot \mathrm{Re}_L)^{9/4}$ in channel flows, where $\mathrm{Re}_L$ is a Reynolds number based on the magnitude of the velocity fluctuations and the integral scale. Ferziger and Peric (1996) report that this Reynolds number is about 1% of the macroscopic Reynolds number, in which case for a Reynolds number of 50,000 the number of points should be about $1.4\times10^7$ ("per large eddy"). In 1995 it was possible to run DNS simulation using about $1.35\times10^8$ points on an Intel Delta parallel computer. Each time step took about 1 min. (Ferziger and Peric, 1996). Using Comte-Bellot (1963) channel flow experiments, Wilcox (1993) calculated that for a Reynolds number of 61,600, corresponding approximately to $1.5\times10^8$ grid points, 63,000 time steps would be required to reach steady state, which would require more than 1,000 hours of CPU time on a 1995 Intel Delta parallel computer. Leschziner (1995) gives a more detailed estimate of the computer resources and corresponding CPU time for DNS.

DNS is therefore impractical for engineering applications so far, and is limited to small Reynolds numbers. Nevertheless, it has proved invaluable in supplying time-dependent information related to the flow variables at a large number of grid points. These results, most of which are very difficult to measure experimentally (e.g. pressure term), can be regarded as "numerical experimental data". These are used to produce statistical information that can be used to develop a better qualitative understanding of the flow physics and propose new closure models for the RANS.

### 2.3.2 Large Eddy Simulation (LES)

The large scale motions are generally more energetic than the small scales and their size and strength make them better vectors of transport affecting the mean flow. LES provides a way of combining the accuracy of DNS to treat the large scale eddies with the efficiency of some form of RANS equations to treat the smaller dissipative turbulence scales.

A filter function (e.g. Gaussian, box, cutoff, Fourier) is used to distinguish eddies that are going to be calculated from those that are going to be modelled. Each filter has associated with it a length scale, $\Delta$, which represents the threshold below which eddies will be modelled. Above $\Delta$ DNS technique applies. Below $\Delta$ a space-averaged version of the Navier-Stokes equations is produced, which is very similar to equation (2.1). Instead of the turbulent Reynolds stresses, similar terms (but space-averaged) called Sub-Grid Scale (SGS) Reynolds stresses need to be modelled so that the problem can be solved. Several models exist such as the Smagorinsky (1963) eddy viscosity model or the dynamic model of Germano *et al.* (1990), to quote just two. The interested reader is referred to Ferziger (1995) for a review.

With LES a much coarser mesh and larger time steps can be used for the calculation of a given flow problem. Furthermore, as the size of the smallest eddies decreases with increasing Reynolds number it is possible, for a given computing cost, to achieve simulation of much higher Reynolds number flows with LES than DNS. Wilcox (1993) reports that the ratio between the number of points required for LES and the number of points required for a DNS simulation of a channel flow problem is about $\left(0.4/Re_L^{\ 1/4}\right)$ i.e. 8.5% using a Reynolds number of 50,000. Consequently LES is preferred whenever the Reynolds number is too high or the geometry too complex for the application of DNS. Promising work has been published in recent years applying LES to open-channel flows similar to the FCF (Thomas and Williams, 1995a and 1995b, Shi *et al.*, 2000) and to wind engineering problems (Easom, 2000). Yet, LES is very resource intensive, as both previous references demonstrate. Consequently it still remains a research tool.

One LES parent method is called the Coherent Structure Capturing (CSC) or Very Large Eddy Simulation (VLES). It is used to calculate high-energy large coherent structures in shear flows. The specific focus on large structures allows an increase in size of the LES filter, to obtain a much coarser grid than with the standard LES (about 1% of the standard number of grid points required for LES; Ferziger, 1993).

### 2.3.3 Chaos

The link between chaos and turbulence is explained by Lesieur (1987), on the basis that coherent structures can be generated by (random) fluctuations in the flow. In that sense one can say that turbulence has emerged from a perturbation of chaotic nature. As a result there is a physical correlation between perturbation approaches and turbulence, which justifies the application of chaos theory to turbulence. This notion is exemplified in the classic illustration related to the sensitivity of a chaotic system to initial conditions and known as the "butterfly effect". This type of sensitivity to the initial conditions has also been observed in DNS and LES models (Wilcox, 1993), which would confirm the similarity between chaos and turbulence.

The application of chaos theory to turbulence also relies on the mathematical foundation of this theory, which treats non-linear dynamic systems that are chaotic in time, and on its ability to represent simple turbulent effects such as the Rayleigh-Bénard convection (Wilcox, 1993). Recent progress in the mathematical field has indicated that statistical turbulence features can be formulated in a way that meets chaos theory requirements. Hence a chaotic model of complex turbulent flows based on the turbulence spectrum should be feasible. Nevertheless both Lesieur's (1987) and Wilcox's (1993) writings reflect the difficulty in formulating the turbulence problem in a solvable manner. In particular Wilcox reports that the spectrum of wavelength is so large in a turbulent problem that it is far greater than that of the dynamical systems that have been studied.

Chaos description of turbulence clearly is in its infancy and far from being applicable to practical cases. However its value lies in developing a better qualitative understanding of turbulence initiation and growth. Quoting Bradshaw Wilcox (1993) also demonstrates that simple chaos models would probably be as resource intensive as LES.

### 2.3.4 The Discrete Vortex Method

This method seems a much more viable alternative to the RANS than chaos theory. Very generally, the method assumes an assembly of vortices of a given mathematical form (e.g. point vortices). The interaction between these vortices is used to simulate turbulent shear layer. The reader is referred to Easom (2000) for further references.

## 2.4 SUMMARY

This chapter has introduced the concept of turbulent flows and their mathematical formulations. As previously stated this thesis will use the RANS equation formulation closed by a suitable turbulence model. Particular attention was devoted to the presentation of the standard $k$-$\varepsilon$ model and RSM, for their respective popularity and completeness. They will be used later to investigate the role of turbulence in two-stage channel hydrodynamics. The final section gave an indication of what is to come in the future of turbulence research and long-term applications. In particular, LES looks very promising in the medium term to solve localised engineering problems in detail, although it requires significant operator skills and powerful hardware (Easom, 2000).

## 2.5 REFERENCES FOR CHAPTER 2


1. Alfrink, B.J., (1982), "Value of Refined Turbulence Modelling for the Flow over a Trench", Proc. Symp. On Refined Modelling of Flows.
2. Basara, B., Cokljat, D., (1995), "Reynolds-Stress Modeling of Turbulent Flows in Meandering Channels", ASME, Fluid Engineering Division (FED), Vol. 221, pp. 27-32.
3. CFX, (1997), "CFX 4-2: Solver Manual", AEA Technology plc, UK.
4. Chou, P.Y., (1945), "On Velocity Correlations and Solutions of the Equations of Turbulent Fluctuations", Quart. Appl. Math., Vol. 3, p. 38.
5. Cokljat, D., Younis, B.A., (1995), "Second-order Closure Study of Open-Channel Flows", J. Hydr. Eng., Vol. 121, No. 2, pp. 94-107.
6. Cokljat, D., (1993), "Turbulence Models for Non-circular Ducts and Channels", PhD Thesis, City University, London, UK.
7. Darly, B.J., Harlow, F.H., (1970), "Transport Equations in Turbulence", Phys. Fluids, Vol. 13, p. 2634.
8. Demuren, A.O., (1993), "A Numerical Model for Flow in Meandering Channels with Natural bed Topography", Water Resources Res., Vol. 29, No. 4, pp. 1269-1277.
9. Easom, G., (2000), "Improved Turbulence Models for Computational Wind Engineering", PhD Thesis, The University of Nottingham, UK.
10. Ferziger, J.H., Peric, M., (1996), "Computational Methods for Fluid Dynamics", Springer-Verlag, Berlin.
11. Ferziger, J.H., (1995), "Large Eddy Simulation", in "Simulation and Modelling of Turbulent Flows", M.Y. Hussaini and T. Gatski (Eds.), Cambridge Univ. Press, New York.
12. Ferziger, J.H., (1993), "Estimation and Reduction of Numerical Error", presented at ASME Winter Annual Meeting, Washington, quoted in Ferziger and Peric (1996).

13. Germano, M., Piomelli, U., Moin, P., Cabot, W.H., (1990), "A Dynamic Subgrid Scale Eddy Viscosity Model", Proc. Summer Workshop, Centre for Turbulence Research, Stanford, California.
14. Hanjalic, K., Launder, B.E., (1972a), "A Reynolds Stress Model of Turbulence and its Application to Thin Shear Flows", J. Fluid Mech., Vol. 52, p. 609.
15. Hanjalic, K., Launder, B.E., (1972b), "Fully Developed Assymetric flow in a Plane Channel", J. Fluid Mech., Vol. 51, p. 301.
16. Janin, J.M., Marcos, F., Denot, T., (1997), "Code TELEMAC-3D Version 2.2 – Note Théorique", Report HE-42/97/049/B, EDF-DER, LNH Chatou, France (In French).
17. Launder, B.E., Reece, G.J., Rodi, W., (1975), " Progress in the Development of a Reynolds Stress Turbulence Closure", J. Fluid Mech., Vol. 68, p. 537.
18. Launder, B.E., Li, S.-P., (1994), "On the Elimination of Wall-Topography Parameters from Second-Moment Closure", Phys. Fluids, Vol. 6, No. 2, pp. 999-1006.
19. Leschziner, M.A., (1995), "Modelling Turbulence in Physically Complex Flows", HYDRA 2000 (Vol. 2), Thomas Telford, London.
20. Lesieur, M., (1987), "Turbulence in Fluids", Nijhoff Publishers (Kluwer Academic Publishers Group).
21. Lin, B., Shiono, K., (1995), "Numerical Modelling of Solute Transport in Compound Channel Flows", J. Hydr. Res., Vol. 33, No. 6, pp. 773-788.
22. Manson, J.R., (1994), "The Development of Predictive Procedure for localised Three Dimensional River Flows", PhD Thesis, The University of Glasgow.
23. Mansour *et al.*, (1988)
24. Naot, D., Rodi, W., (1982), "Calculation of Secondary Currents in Channel Flow", J. Hydr. Div., Vol. 108, No. HY8, pp. 948-968.
25. Rodi, W., (1980), "Turbulence Models and their Application in Hydraulics", IAHR State of the Art Report, IAHR.
26. Rotta, J.C., (1951), "Statistiche Theorie Nichthomogener Turbulenz", Z. Phys., Vol. 129, p. 547.
27. Sauvaget, P., Usseglio-Polatera, J.M., (1987), "Numerical Simulation of Stratified Flows in Estuaries and Reservoirs", Proc. Third Int. Symp. On Stratified Flows, Pasadena, California.

28. Schwarz, W.R., Bradshaw, P., (1994), "Term-by-Term Tests of Stress-Transport Turbulence Models in a Three-Dimensional Boundary Layer", Phys. of Fluids, No. 6, pp. 986-998.
29. Shi, J., Thomas, T.G., Williams, J.J.R., (2000), "Free-Surface Effects in Open Channel Flow at Moderate Froude and Reynolds Numbers", J. Hydr. Res., Vol. 38, No. 6, pp. 465-474.
30. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.
31. Smagorinski, J., (1963), "General Circulation Experiments with the Primitive Equations", Monthly Weather Review, Vol. 91, No. 3, p. 99.
32. Speziale, C.G., Sarkar, S., Gatski, T.B., (1991), "Modelling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach", J. Fluid Mech., Vol. 227, pp. 245-272.
33. Stansby, P.K., Zhou, J.G., (1998), "Shallow-Water Non-Hydrostatic Pressure: 2D Vertical Plane Problems", Int. J. Numer. Meth. Fluids, Vol. 28, pp. 541-563.
34. Tennekes, H., Lumley, J.L., (1972), "A Finite Course in Turbulence", The MIT Press, Cambridge, Massahusetts.
35. Thomas, T.G., Williams, J.J.R., (1995a), "Large Eddy Simulation of a Symmetric Trapezoidal Channel at a Reynolds Number of 430,000", J. Hydr. Res., Vol. 33, No. 6, pp. 825-842
36. Thomas, T.G., Williams, J.J.R., (1995b), "Large Eddy Simulation of Turbulent Flow in an Asymmetric Compound Open-Channel", J. Hydr. Res., Vol. 33, No. 1, pp.27-41.
37. Wilcox, D.C., (1993), "Turbulence Modelling for CFD", DCW Industries, Inc., La Cañada, California.
38. Younis, B.A., Gatski, T.B., Speziale, C.G., (1999), "Towards a Rational Model for the Triple Velocity Correlations of Turbulence", NASA Report NASA/TM-1999-209134.

# Chapter 3:

# Literature Review

# Chapter 3:

# Literature Review

*"The gods did not reveal from the beginning all things to us; but in the course of time. Through seeking, men find that which is better".*

(Xenophanes of Colophon; 570 BC - 480 BC)

## 3.1 INTRODUCTION

This chapter will review numerical modelling, and more specifically illustrate how the equations presented in the previous chapter have been applied to solve civil engineering problems.

A brief introduction to numerical modelling in hydraulic engineering is given first, followed by a review of one-, two- and (quasi-) three-dimensional numerical modelling. In this section, in particular, river and coastal engineering applications of the St Venant (1-D) and shallow water (2-D) equations are presented.

A more detailed fluid mechanics section called Computational Fluid Dynamics (CFD) follows. In this section a complete treatment of the full Navier-Stokes equations, as presented in Chapter 2, is considered. A review of past mechanical and civil engineering research work is also given, and the present research work put into perspective in this context.

## 3.2 NUMERICAL MODELLING IN CIVIL ENGINEERING HYDRAULICS

The application of numerical modelling of open channel flows is fairly recent in the course of river engineering history. This can be attributed, in part, to conservatism in the civil engineering design community and the relatively slow uptake of computer applications as a means of problem solving in the industry. Another factor to account for the slow introduction of numerical hydraulics and fluid mechanics in civil engineering is the complexity of the problem and the level of uncertainty in defining boundary conditions. First of all, large scales need to be modelled in river engineering (kilometric scale). Then the complexity of the topography needs to be represented in the model, upstream/downstream and wall boundaries need to be defined and, finally, the problem of the free surface and its fluctuations also needs to be resolved. In a few words the domain to be considered is large, variable in all dimensions and dependent on the flow and boundary conditions. All these considerations add to the mathematical difficulty in solving the non-linear Navier-Stokes equations that describe the physics of the flow.

This resulted in civil engineering taking an initially different route from the other engineering disciplines to enable the implementation of numerical models of river flows. While the other disciplines were able to adopt fluid mechanics treatment of the Navier-Stokes equations in simple small-scale geometries, such as car bodies or aeroplane wings, civil engineering had to formulate the Navier-Stokes equations in a simpler way that would take into account the variability and complexity of the natural solution domain. Moreover, of concern to most civil engineers were the predictions or water levels and discharges to be used for the design of flood protection work, the construction of structures and the assessment of flood risk. These did not require the use of detailed fluid mechanics, but rather the ability to present a large-scale picture of the flow.

### 3.2.1 One-Dimensional (1-D) Modelling

The origin of a mathematical formulation for the flow is related to the work of St Venant, who formulated the unsteady flow equations describing the translation of a flood wave along a river channel in 1871. His mathematical model is based on two partial differential equations accounting for mass and momentum conservation in the physical system:

$$B\frac{\partial h}{\partial t}+\frac{\partial Q}{\partial x}=q_s \tag{3.1}$$

$$\frac{\partial Q}{\partial t}+\frac{\partial}{\partial x}\left[\frac{\beta Q^2}{A}\right]+gA\frac{\partial h}{\partial x}+gA\frac{Q|Q|}{K^2}=0 \tag{3.2}$$

Where the variables are, $Q$ = the channel discharge, $q_s$ = source term (inflow), $x$ = the distance along the channel centreline, $t$ = time, and $h$ the corresponding water surface elevation. The cross-sectional parameters are: $B$ = channel width, $\beta$ = momentum correction factor, $A$ = the cross-section wet area, $K$ = the channel conveyance and $g$ = the Earth's gravitational acceleration.

The underlying assumptions for the above set of equations are:

(i) The flow is one dimensional, i.e. the velocity is uniform over any channel cross-section and the corresponding water level is horizontal;

(ii) The streamline curvature is small and vertical accelerations negligible (hydrostatic pressure);

(iii) The effects of boundary friction and turbulence can be accounted for through resistance laws analogous to those used for steady state flow;

(iv) The average channel slope $\theta$ is small, i.e. $\sin\theta \approx \theta$ and $\cos\theta \approx 1$.

Because of their non-linear nature, the St Venant equations cannot be solved analytically and therefore require being discretised and solved using a numerical method. The most commonly applied method in river modelling is the finite difference technique, although finite volume or finite element methods could equally be used. In such a problem the channel is discretised in different reaches, all of which are accounted for by a single representative cross-section.

One-dimensional numerical models originate from the early work by Isaacson, Stoker and Troesh who modelled and run a mathematical model of portions of the Ohio and Mississipi rivers in 1954. Rapid progress was made in the numerical treatment (Preissman scheme, 1961) and increasing computer resources allowed large scale numerical models to be implemented by consulting firms such as SOGREAH[1] for the Mekong delta and the River Senegal in the 1960s and 1970s (Cunge, Holly and Verwey, 1980). In the U.K., Sir William Halcrow and Partners, Mott Mac Donald, Babtie Shaw and Morton and the Hydraulics Research Station all developed their own river modelling codes. In the United States, the U.S. Army Corp of Civil Engineers also produced the HEC-RAS suite of programmes, which are widely used in engineering courses and practice. Such 1-D models are widely used by the river modelling industry, in particular to determine the impact of extreme flood events.

### 3.2.2 Two-Dimensional (2-D) Modelling

The need to investigate coastal and estuarine hydraulics, where the flow is no longer one dimensional, led research institutions such as the Danish Hydraulics Institute, the US Army Corps of Engineers and EDF-DER (France), to develop 2-D numerical models based on the depth-averaged Navier-Stokes equations. The latter, which are also known as the shallow water equations, can be written as:

$$\frac{\partial h}{\partial t} + \nabla \cdot (\vec{Q}) = q_s \tag{3.3}$$

$$\frac{\partial \vec{Q}}{\partial t} + \nabla \cdot (\vec{U} \cdot \vec{Q}) = -hg\frac{\partial Z}{\partial t} + \nabla \cdot \left(\nu_e h \cdot \nabla \vec{U}\right) + h \cdot \vec{F} \tag{3.4}$$

Where, $\vec{Q} = \begin{bmatrix} Q_x \\ Q_y \end{bmatrix} = \begin{bmatrix} h \cdot U_x \\ h \cdot U_y \end{bmatrix}$, $\vec{U} = \begin{bmatrix} U_x \\ U_y \end{bmatrix}$, $h$ = water depth, $\nu_e = \nu + \nu_t =$ effective kinematic viscosity, sum of the kinematic and turbulence viscosity, $\vec{F} =$ Body and Friction Forces $= \begin{bmatrix} F_x \\ F_y \end{bmatrix}$ and Z the position of the free surface.

[1] Société Grenobloise d'Etude et d'Applications Hydrauliques, France.

The underlying assumptions are:

(i) The wave length is large in relation to the depth of the flow;

(ii) The flow is two-dimensional, i.e. the velocity is uniform over the depth;

(iii) The absence of vertical velocity, i.e. the pressure is hydrostatic;

(iv) The effects of boundary friction can be accounted for through resistance law similar to those used in 1-D.

The computational effort required to carry out such modelling is significantly higher than in one dimension, but resulting simulations are able to represent flow events such as the arrival and extent of a flood, dam break and flow separation (recirculation and dead zones). The first models used finite difference techniques (Leendertsee, 1967; Vreugdenhil and Wijbenga, 1982) and the method of characteristics (Towmson, 1974). With the development of the finite element method (Brebbia *et al.*, 1978; Wang *et al.*, 1985) and associated numerical techniques (Brookes and Hughes, 1982; Hervouet, 1992), enhanced hydraulics codes were produced, such as RMA-2 (King and Norton, 1978) in the early 1970s and TELEMAC-2D in the late 1980s (Hervouet, 1991). The finite element method proved useful in representing complex geometries; however it is a demanding method to implement numerically and has been associated with mass-conservation difficulties. A hybrid of both finite difference and finite element methods called the finite volume method emerged, originally to solve the full Navier-Stokes equations (Patankar and Spalding, 1972; Demirdzic *et al.*, 1987; Karki and Patankar, 1988). It is conservative, numerically accurate, simple and has been applied to a few two- and quasi three-dimensional flow problems (Lai and Yen, 1992).

Extensive literature is available regarding the application and usefulness of two-dimensional analysis. Early work specifically focussed on lake or coastal applications where a two-dimensional treatment was necessary. Platzmann (1958) investigated surge storms in lakes for example. Kuipers and Vreugdenhil (1973) used a shallow-water approach to investigate a variety of steady state problems, including river channels. Mc Guirk and Rodi (1978) and Lean and Weare (1979) used two-dimensional analysis to reproduce recirculation effects in estuaries and harbours, and Falconer (1984) for water quality prediction in tidal embankments.

More recently the focus has extended to flood events and dam break problems, in particular after considerable work related the treatment of wetting-and-drying and adaptive meshing was carried out (Lynch and Gray, 1980; Akanbi and Katopodes, 1988; Molinaro and Natale, 1994; Tchamen and Kawahita, 1998). In particular the work by Bates *et al.* (1993, 1996) from Bristol University has illustrated the capability of the finite element code TELEMAC-2D to reproduce the transit of a flood wave and the corresponding flood map dynamically. Such codes can now support river models of up to 60 km (Bates *et al.*, 1996). Other researchers such as Wijbenga (1985), Akanbi and Katopodes (1988), King and Roig (1991), Paquier and Farissier (1997), Sleigh *et al.* (1998), Markhanov *et al.* (1999) have also documented the use of a two-dimensional approach for flood modelling. These papers have mostly focussed on the dynamic flooding in the plan view, in order to reproduce the spreading of the flood flow, for which two-dimensional codes are perfectly suited.

Hervouet and Rouge (1996) and Zoppou and Roberts (1999) used the shallow water equations to reproduce the catastrophic collapse of dams and water supply reservoirs. In the work by Hervouet and Rouge in particular, a real case scenario from 1959 in the South of France (Malpasset dam collapse near Fréjus) was successfully reproduced: The computed flood wave advance was compared thanks to recorded data.

One handicap to the installation of such codes in industry could be related to the difficulty of implementing suitable grids on large areas using workstations (Hervouet and Janin, 1994), and the general operating cost. However, as the cost of computer power is decreasing rapidly, it is the author's view that such codes will become more common in civil engineering applications within the next few years. In general their application requires a greater level of understanding of principles and model limitations than one-dimensional codes.

### 3.2.3 Three-dimensional (3-D) Modelling

Three-dimensional codes for civil engineering applications are a relatively recent innovation. In fact, TRISULA (Delft Hydraulics, The Netherlands) MIKE 3 (Danish Hydraulics Institute) and TELEMAC-3D (EDF-DER, France) are probably the first such commercial packages available on the market. These however are not really three-dimensional codes as they rely heavily on the hydrostatic assumption to yield a solution in a layer-averaged format (i.e. the vertical dimension is a series of two-dimensional solutions). Vertical velocities are calculated from the conservation of mass. They resemble two-dimensional codes on which they have often been based. As such they are unable to account for complex three-dimensional flow features or detailed turbulence and can be referred to as quasi three-dimensional codes. These codes have not been designed for detailed three-dimensional flows with high vertical velocities. In fact their application has been in the field of coastal and ocean engineering where a layered approach is sufficient to account for the relatively low vertical velocities. Their objective is to represent horizontal currents as well as salinity or temperature gradients in the water column (Peltier *et al.*, 1996; Gross *et al.*, 1999). Strong vertical flow or pressure-driven features in rivers should therefore not be well reproduced.

As underlined above the application of quasi three-dimensional models is very similar to that of the two-dimensional shallow water models, with probably more emphasis on ocean and coastal applications for the treatment of salinity, pollution or wind effects over the depth. Leendertsee (1973), Blumberg and Mellor (1983), Hall *et al.* (1992), Peltier *et al.* (1999) have applied this type of model to computing flow, water quality and/or sediment transport in seas and estuaries. Some applications have also dealt with recirculation in lakes or bays (Liggett, 1969; Koutitas and O'Connor, 1980; Falconer *et al.*, 1991). Some attempts were made to reproduce velocity fields in rivers (Benqué *et al.*, 1982; Blumberg *et al.*, 1990), and even flood flows (Ammer and Valentin, 1993) with mixed success however. Lavedrine (1996, 1997) attempted to reproduce a Flood Channel Facility test case, but only presented brief results. In cases where sediment transport was investigated in channels, the hydrostatic pressure assumption clearly showed its limitations, because it fails to reproduce pressure-driven recirculation (Shimizu *et al.*, 1990), and the flow concentration at the inflection point of a bend in a large depth-ratio, flooded channel (Fukuoka and Wanatabe, 2000).

The interest of trying to implement fully three-dimensional models for natural open-channels has been recently discussed by Knight (1996), Neary *et al.* (1999) and Bettess and Fisher (1999). It has long been known that flow features such as turbulence or the flow dynamics at bends or in flooded rivers are three-dimensional. Recent results regarding sediment transport have also indicated the importance of pressure (Fukuoka and Wanatabe, 2000). Yet little research work has been undertaken regarding the implementation of fully three-dimensional modelling techniques to river flows (Sinha *et al.*, 1998), and all flood flow models have involved simple prismatic channels (Krishnappan and Lau, 1986; Morvan *et al.*, 2000). Probably because there was no demand for such detailed investigation in civil engineering, it seemed very expensive, and the implementation of such refined technique seemed in contradiction with the uncertainty in defining the parameters necessary to describe a river's behaviour. This means that all fully three-dimensional numerical codes available today have been developed for applications in other branches of engineering, where their benefits could be more easily realised (Anderson, 1995). This new interest has been mostly motivated from a research point of view to better understand the hydrodynamics and turbulence mechanisms in open-channel flows. Some recent reports from industry (Bettess and Fisher, 1999) and academia (Swindale, 1999) have indicated that three-dimensional modelling could be very useful in river restoration projects for biological and morphological investigations.

### 3.2.4 Summary

This section has discussed the application of numerical modelling to river flood simulation in civil engineering. The scale and the complexity of river systems led to the initial development of simple hydraulics models required for design purposes. In particular, straightforward simplifications have occurred regarding the representation of the pressure field and turbulence. This has been possible because the flow features of interest to the users were mostly happening in the plan view and at macroscopic scales. Such models have been extensively used in industry for the forecast of flood extent (1-D and 2-D), flood flow transit (2-D), coastal hydraulics (2-D and quasi 3-D) or sediment and pollutant transport (1-D, 2-D and quasi 3-D), and the design of adequate engineering solutions.

On the other hand there has been little use of three-dimensional fluid dynamics models in civil engineering; mostly because industry is uncertain as to their capability and because the data required for their construction and implementation make them difficult and costly to apply.

## 3.3 COMPUTATIONAL FLUID DYNAMICS (CFD)

This section aims to reviewing some recent applications of the full Navier-Stokes equations to environmental flow problems. This flow simulation method has become a subject of its own, and is usually referred to as Computational Fluid Dynamics (CFD). As discussed previously its roots are in aeronautical and mechanical engineering in the mid-1950s, originally for national defence applications such as ballistic missiles and aeronautical design, space exploration and the development of nuclear power.

### 3.3.1 What is CFD?

> *"Computational Fluid Dynamics is, in part, the art of replacing the governing partial differential equations of fluid flow with numbers, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow field of interest."*
>
> (Anderson, 1995)

Such a simple definition can also be found in Versteeg and Malalasekera (1996) and does not differentiate CFD from the other types of numerical modelling previously presented. The definition of CFD lies in its mechanical engineering background. CFD offers the most general treatment of the Navier-Stokes equations and can be described as a fluid mechanics treatment of the Navier-Stokes equations in which pressure and turbulence, in particular, are fully accounted for. Numerical modelling as previously described for civil engineering applications has so far been a simplified application of CFD, mostly limited to a vertically averaged treatment of the Navier Stokes equations, assuming a hydrostatic pressure distribution.

The definition of CFD is also related to its domain of application and its contribution to the design of industrial products and processes in aeronautics, the car industry, naval architecture, marine engineering, power generation and chemical engineering. Its philosophy is in fact that of a mechanical Computer-Aided Engineering (CAE) tool, with a view to the efficient design and manufacture of industrial parts. It lies in the need for a detailed fluid mechanics solution of the Navier Stokes equations, where the effects of turbulence and pressure can be significant (e.g. in the design of a wing that optimises an aeroplane uplift and reduce fuel consumption). Its development was supported by the tremendous drive provided by governments and industrial firms to design rockets, planes, jet engines, turbines, turbomachines, furnaces and nuclear reactors. As illustrated by Anderson (1995) for example, CFD was the key to the solution of the blunt body problem in the 1960s, a solution reflected in the design of the Mercury and Apollo space capsules. These industries heavily rely on CFD for their design, and this technique is often substituted to physical experiments.

To summarise CFD was developed to investigate the design of manufactured shapes that would allow optimum fluid flow or mixing process. It is a general fluid mechanics approach aimed at representing complex three-dimensional flows.

### 3.3.2 CFD Applied to Open-Channels

CFD has only had limited application to the simulation of open-channel flow problems. Early attempts at simulating open channel flow treated the channel as a modified duct problem (Rastogi and Rodi, 1978; Naot and Rodi, 1982; Gibson and Rodi, 1989). Therefore the mechanical engineering roots of CFD is clearly evident in these publications.

A few recent publications however (Demuren, 1993; Cokljat and Younis, 1995 and Basara and Cokljat, 1995; Sinha *et al.*, 1998; Hodskinson and Ferguson, 1998; Lane *et al.*, 1999; Mesehle and Sotiropoulos, 2000; Wu *et al.*, 2000; Nguyen *et al.*, 2000; Bradbrook *et al.*, 2000)) have reflected an increasing interest in applying CFD to civil and environmental channel flow problems. Yet to the author's knowledge, Sinha *et al.* (1998) are one of the few only groups to have thoroughly treated a full-scale river problem three-dimensionally, for what was a relatively simple plan-view layout; Demuren (1993), and Basara and Cokljat (1995) are among the few researchers to have treated a meandering channel of prismatic cross-section. A brief review of various three-dimensional numerical models is presented in Table 3.1. This is by no means an exhaustive list, but is rather an historical review of the most significant contributions to modelling open-channels with CFD. A few of the listed papers, with potential direct application in different fields of river engineering, are detailed below.

Demuren (1993) applied CFD techniques to an inbank flow problem in a meandering channel. He reproduced the physical experiment of Almquist and Holley (1985), who investigated the bed shear stress and depth averaged velocity field in a meandering channel with natural bed topography. The numerical method that he used is summarised in Table 3.1. His results show a reasonable comparison with experimental data, however the calculated bed shear stresses are overestimated and miss out localised detail. This could be due to an insufficient level of numerical discretization as the author used a relatively coarse grid and did not carry out mesh independence tests. In addition, the use of a $k$-$\varepsilon$ turbulence model could also justify some inconsistencies as it is known to perform poorly in flows over curved surfaces or with flow separation. On the other hand, from a practical viewpoint, these results are encouraging as they present an efficient method that would provide a satisfactory level of accuracy to calculate the flow and sediment pattern in a natural channel.

Cokljat and Younis (1995) presented the results of a detailed investigation of the flow and turbulence fields in straight channels. The authors used numerical results reported by Nezu and Rodi (1985) and Tominaga *et al.* (1989) to assess the quality of their model. The focus is on the need to accurately represent the anisotropic nature of turbulence in addition to the free-surface effects since these are responsible for the weak recirculations evident in straight prismatic channels. This research however remains removed from practical application since it requires fine case-specific numerical developments, in particular regarding turbulence modelling, although it clearly attempts to investigate open-channel hydraulics. Associate papers report interesting findings regarding the accuracy obtained using a simple $k$-$\varepsilon$ model versus a full RSM for a flow over a trench (Basara and Younis, 1995) or in meandering channels (Basara and Cokljat, 1995). This work follows up on the physical experiments carried out at HR Wallingford into the Flood Channel Facility concerning straight inbank and overbank flows (Series A) and is very important for the detailed work carried out on the modelling of turbulence. Similar work was also published at the same period by Lin and Shiono (1995) using a non-linear $k$-$\varepsilon$ turbulence model and Thomas and Williams (1995a, b) using LES.

Sinha *et al.* (1998) is the most interesting paper from a practical point of view as it reports on the modelling of a full-scale reach for a natural river of simple plan form. These authors investigated the flow hydrodynamics downstream of the Wanapum Dam through a 4-km reach of the Columbia River. Their main concern was to determine the effects of the hydropower installations on the aquatic ecosystem and the habitat provision, downstream of the dam. The details of the numerical model are given in Table 3.1. With practical considerations in mind, the authors implemented a simple $k$-$\varepsilon$ model and a two-point wall function to simulate roughness at the walls to ensure a good representation of roughness and facilitate calibration. The results show good agreement between numerical and field data for general velocities at a given relative depth. A detailed investigation on the effects of roughness is also presented, which demonstrates the flexibility of the numerical model and relates this to river engineering. Unfortunately this model is only applied to one river reach, which makes it difficult to fully assess the code's quality. Yet, as underlined in the paper, "*this study is the first comprehensive attempt to account numerically for most of the complexities encountered in natural river geometries*". At the same period established geomorphologists such as Lane and Ferguson published results of CFD applications to small-scale river bends and confluences, although with less emphasis on the numeric (Hodskinson and Ferguson, 1998; Lane *et al.*, 1999, Bradbrook *et al.*, 2000). The scale of their problems is usually very small (smaller than the FCF in plan view), which is likely to make them very dependent on the boundary conditions unfortunately. These publications however illustrate the usefulness and potential of three-dimensional CFD to model recirculation, mixing processes and bed shear stress accurately.

Meselhe and Sotiropoulos (2000) presented early results obtained from their own investigation of open-channels using CFD. A simple numerical technique is used, as displayed in Table 3.1 and an Alternate Direction Implicit (ADI) method is implemented for the solution of pressure. A two-point wall function is also used at the wall. The merit of their investigation is an attempt to account for the variation of the free-surface using previous time-step pressure information at the surface boundary. Such an approach should also be available in commercial codes such as CFX (v. 4.4). This means that it should be possible to investigate time-dependent flow conditions such as floods using fully three-dimensional models. Mesehle and Sotiropoulos tested their model against experimental data sets for a meandering inbank flow flume (Yen, 1965) and a bend (Rozovskii, 1957). Early results seem encouraging, although few details are provided. What is noticeable is that the use of the free-surface algorithm does not seem to enhance the quality of the solution compared to the rigid lid approach; however the solution presented used an imperfect set of turbulence equations, the latter not being fully coupled with the free-surface algorithm. The authors are well aware of such insufficiencies as underlined in their paper. In their conclusions they mention the need to improve the solution technique by implementing a multigrid solver, and to investigate anisotropic turbulence modelling before moving on to modelling natural river reaches.

A recent contribution by Wu *et al.* (2000) has confirmed the rising interest from the civil engineering community for CFD techniques applied to rivers. This paper presents the results of a fully three-dimensional application in which a time dependent treatment of the free surface as well as a sediment transport model are included. The focus is on sediment transport. The model uses a simple $k$-$\varepsilon$ model and the Stone (1968) method for the resolution of the discretized equations. Further detailed are showed in Table 3.1. The free surface is using outputs from a two-dimensional solution to update the mesh, an approach also used by the TELEMAC system. The sediment transport equations are based on the classic van Rijn formula (1987). The later model is tested in fairly simple conditions and show good agreements with observations for the main morphological and sediment transport features. Detailed analysis reveals localised discrepancies that most probably stems from turbulence and from the sediment transport model, where the background science is much less accurate. Since no detailed hydrodynamics results are presented this is left to speculation however. This paper is interesting because it offers the prospect that fully three-dimensional sediment transport features could be finally investigated and maybe resolved numerically.

Civil and environmental engineering applications of CFD have still to demonstrate their full potential in terms of applicability, efficiency and accuracy (i.e. does it correctly produce data of use to civil engineers?). Different elements of fundamental importance are currently being investigated by several research groups worldwide. The above papers have shown that in the recent past significant progress has been made in applying CFD to river engineering with:

(i) the application of CFD to flumes in meandering inbank channels (Demuren, 1993);

(ii) the investigation of turbulence modelling in open-channel (Younis research group, 1995, Lin and Shiono, 1995, Thomas and Williams, 1995a, b);

(iii) the application of CFD to full-scale inbank flow problems (Sinha *et al.*, 1998; Hodskinson and Ferguson, 1998; Lane *et al.*, 1999; Bradbrook *et al.*, 2000));

(iv) the representation of the free surface dynamically (Mesehle and Sotiropoulos, 2000; Wu *et al.*, 2000);

(v) and the inclusion of sediment transport routines (Wu *et al.* 2000).

### 3.3.3 CFD Applied to other Civil and Environmental Problems

In the field of water engineering CFX has also been used in the design of wastewater processing tanks (see CFX brochures). This is because an accurate treatment of complex but weak recirculations was necessary to ensure that a proper mixing was happening in the tank. At the University of Glasgow it has also been suggested to use CFD in the design of fish ladders up dam structures, to verify the level of turbulence and strength of currents in each basin.

A large field of application of CFD of interest to environmentalists and civil engineers is wind engineering. Easom's recent thesis (2000) illustrates the application of CFD to investigate the wind loading on building with respect to the current design practise. It shows the benefits such technique could have on improving the wind flow around buildings to enhance their design. The author believes that such an application could also be fundamental regarding the modelling of air quality in high building environments, or the wind flow around large structures such as long-span bridges.

Ventilation is another application related to building engineering. The modelling of ventilation and air circulation in offices and other work places is already actively carried out in Sweden for example (Nilsson and Holmberg, 2000). Ventilation in mining shafts is conducted at the School of Chemical, Environmental and Mining Engineering at Nottingham University (Hargreaves *et al.*, 1998). These models are now becoming part of the design process of large building structures in which air circulation is essential to comfort, efficiency and safety. Other applications of CFD are concerned with the transport of so called "chemical entities" in power plants, for example Martin-Valdepenas *et al.* (2000), for obvious safety considerations, or the propagation of fire in buildings or offshore oil rigs (see CFX brochure, 2000). In these cases architects and engineers use CFD as a safety-enhancement tool.

### 3.3.4 Numerical Considerations

An overview of Table 3.1 reveals that, if in a first stage the main concern of the investigators has been to improve the representation of the flow physics (from parabolic to full Reynolds Averaged Navier-Stokes equations, and from $k$-$\varepsilon$ model to RSM and LES), recent attempts have focused on the implementation of more accurate discretization schemes and efficient solution techniques. Hence, QUICK (Sofiadilis and Prinos, 1999) has replaced the Spalding method (a combined Hybrid-CDS) and the Tri-Diagonal Matrix Algorithm (TDMA) has been replaced in recent years by the Alternate Direction Implicit

(ADI) method and its improved successor, Stone's method (Stone, 1968). Several publications (Anderson, 1995; Mesehle and Sotiropoulos, 2000) have indicated that the future of CFD applied to complex turbulent flow problems would probably lie in even more efficient numerical solvers such as the multigrid method. The latter enhances convergence where the ADI and Stone's method can fail (especially when the grid structure is complex and lacks order).

These improvements are important since a significant part of the difficulty of modelling natural channel flows is related to the size and complexity of the numerical domain.

### 3.3.5 Summary and Position of the Proposed Research

This chapter has presented Computational Fluid Dynamics in its original mechanical engineering context, but has also illustrated the current research and potential of this field to civil and environmental engineering applications.

It is obvious that the scarcity of papers and the multitude of subjects treated reveal that the application of CFD to open-channel flow problems is still in its infancy. In particular no or few applications have been made that investigate detailed velocity and turbulence measurements in natural meandering channels for flood flows. This is what the current thesis aims to achieve. It is therefore positioned in the continuity of the work presented in the literature review (Table 3.1).

In recent years, a certain consensus has been found in the CFD community regarding the requirements of open-channel models. This means that sound models have been developed, and consideration has been given to both practical and more exact models. In general, skilled users will be able to find documentation detailing the capacity and shortcomings of different formulations for the flow and turbulence equations. This does not imply that research is not being pursued to improve the models, but that material has been produced to describe most fluid mechanics problems satisfactorily, and that it requires testing. As the need for accuracy has increased and the size of problems has grown, users have expressed the need for higher order discretization schemes and, more efficient and stable numerical solvers. The author's results will make use of the hybrid method, but will also rely on an enhanced version of QUICK when necessary. Stone and multigrid solvers will be used following recent recommendations in the literature. This will also contribute to the course of the current research regarding the application of more efficient solution algorithms in complex flow problems.

## 3.4 REFERENCES FOR CHAPTER 3


1. Akanbi, A.A., Katopodes, N.D., (1988), "Model for Flood Propagation on Initially Dry Land", J. Hydr. Eng., ASCE, Vol. 114, No. 7, pp. 689-707.
2. Almquist, C.W., Holley, E.R., (1985), "Transverse Mixing in Meandering Laboratory Channels with Rectangular and Naturally Varying Cross-Sections", Report CRWR 205, The University of Texas, Austin, Texas, USA.
3. Ammer, M., Valentin, F., (1993), "A Hierarchical Finite Element for Three-Dimensional Free Surface Flows", Proc. 5th Int. Symp. Refined Flow Modelling and Turbulence Measurements, Paris, France, pp. 711-718
4. Anderson, J.D., (1995), "Computational Fluid Dynamics – The Basics with Applications", McGraw-Hill, New York, USA.
5. Basara, B., Cokljat, D., (1995), "Reynolds-Stress Nodelling of Turbulent Flows in Meandering Channels", ASME, Fluids Engin. Division (FED), Vol. 221, pp. 27-32.
6. Bates, P.D., Anderson, M.G., Price, D.A., Hardy, R.J., Smith, C.N., (1996), "Analysis and Development of Hydraulic Models for Floodplain Flows", in "Floodplain Processes", M.G. Anderson, D.E. Walling, P.D. Bates (eds.), Wiley.
7. Bates, P.D., Anderson, M.G., (1993), "A Two-Dimensional Finite Element Model for River Flow Inundation", Proc. R. Soc. London A, Vol. 440, pp. 481-491.
8. Bates, P.D., Horritt, M.S., Smith, C.N., Mason, D., (1997), "Integrating Remote Sensing Observations of Flood Hydrology and Hydraulic Modelling", Hydr. Proces., Vol. 11, pp. 1777-1795.
9. Benqué, J.-P., Haughel, A., Viollet, P.-L, (1982), "Numerical Models in Environmental Fluid Mechanics", in "Engineering Applications of Computational

Hydraulics – Homage to Alexandre Preissmann", Vol. 2, M.B. Abbott and J.A. Cunge (eds.), Pitman

10. Bettess, R., Fisher, K.R., (1999), "Lessons to Learn from the UK River Restoration Projects", River Basin Modelling Management and Flood Mitigation Concuted Action (RIBAMOD), Proc. Sec. Workshop on Impact of Climate Change on Flooding and Sustainable River Management, R. Casale, P. Samuels, A. Bronstert, (eds), (to appear).
11. Blumberg, A.F., Galperin, B., O'Connor, D.J., (1992), "Modelling Vertical Structure of Open-Channel Flows", J. Hydr. Eng., ASCE, Vol. 118, No. 8.
12. Blumberg, A.F., Mellor, G.L., (1983), "Diagnostic and Prognostic Numerical Circulation Studies of the South Atlantic Bight", J. Geophysical Res., Vol. 88, No. C8, pp. 4579-4592.
13. Bradbrook, K.F., Lane, S.N., Richards, K.S., (2000), "Numerical Simulation of three-Dimensional, Time-Averaged flow Structure at River Channel Confluences", Water Res. Resour. Res., Vol. 36, No. 9, pp. 2731-2746.
14. Brebbia, C.A., Gray, W.G., Pinder, G.F., (eds.) (1978), "Finite Elements in Water Resources", Proc. Second Int. Conf., London, UK.
15. Brooks, A.N., Hughes, T.J.R., (1982), "Streamline Upwind/Petrov Galerkin Formulations for Convection Dominated Flows with Particular Emphasis on the Incompressible Navier-Stokes Equations", Comp. Meth. Appl. Mech. and Eng., Vol. 32, pp. 199-259.
16. Cokljat, D., Younis, B.A., (1995), "Compound Channel Flows: A Parametric Study using a Reynolds-Stress Transport Closure", J. Hydr. Res., Vol. 33, No. 3, pp. 307-320.
17. Cunge, J.A., Holly, F.M. Jr., Verwey, A., (1980), "Practical Aspects of Computational River Hydraulics", Pitman.
18. Demirdzic, I., Gosman, A.D., Issa, R.I., Peric, M., (1987), "A Calculation Procedure for Turbulent Flow in Complex Geometries", Comput. Fluids, Vol. 15, No. 3, pp. 251-273.
19. Demuren, A.O., (1993), "A Numerical Model for Flow in Meandering Channels with Natural bed Topography", Water Resources Res., Vol. 29, No. 4, pp. 1269-1277.

20. Easom, G., (2000), "Improved Turbulence Models for Computational wind Engineering", PhD Thesis, The University of Nottingham, UK.
21. Falconer, R.A., (1984), "Temperature Distributions in Tidal Flow Field", J. Env. Eng., ASCE, Vol. 110, No. 6, pp. 1099-1116.
22. Falconer, R.A., George, D.G., Hall, P., (1990), "Three Dimensional Numerical modelling of Wind-Driven Circulation in a Shallow Homogeneous Lake", J. Hydrology, Vol. 124, pp. 59-79.
23. Fukuoka, S., Wanatabe, A., (2000), "Numerical Analysis on Three Dimensional Flow and bed Topography in a Compound Meandering Channel", Proc. 4th Int. Conf. Hydroinformatics, Iowa City, USA (CD-Rom).
24. Gibson, M.M., Rodi, W., (1989), " Simulation of Free Surface Effects on Turbulence with a Reynolds Stress Moedel", J. Hydr. Res., Vol. 27, No. 2, pp. 233-244.
25. Gross, E.S., Koseff, J.R., Monismith, S.G., (1999), Three-Dimensional Salinity Simulations of South San Francisco Bay", J. Hydr. Eng., Vol. 125, No. 11, pp. 1199-1209.
26. Hall, M., Shiono, K, Falconer, R.A., (1992), "Three Dimensional Layer Averaged Model for Tidal Flows", Proc. 2nd Int. Conf. Hydraulic Modelling and Environmental Modelling, Bradford, UK.
27. Hargreaves, D.M., Moloney, K.W., Lowndes, I.S., (1998), "Preliminary Computational Fluid Dynamics (CFD) Simulations of Methane Dispersion in a Heading", Proc. 2nd Int. Symp. On Mine Environmental Engineering, pp. 204-231.
28. Hervouet, J.-M., (1991), "Vectorisation et Simplification des Algorithmes en Eléments Finis", EDF, Bulletin de la Direction des Etudes et Recherches Série C, Mathématiques, Informatiques, No. 1, 1991, pp. 1-37, (in French).
29. Hervouet, J.-M., (1991), "Characteristics and Mass-Conservation. New Developments in TELEMAC-2D", EDF-DER, LNH Chatou, France, Groupe Hydraulique Fluviale, Report HE-43/92-41.
30. Hervouet, J.-M., Janin, J.-M., (1994), "Finite Element Algorithms for Modelling Flood Propagation", in "Modelling Flood Propagation Over Initially Dry Areas", P. Molinaro and L. Natale, ASCE, USA.

31. Hervouet, J.M., Rouge, D.,(1996), "Numerical Simulation of the Malpasset Dam-Break Flood Wave", EDF-DER, LNH Chatou, France, Groupe Hydraulique Fluviale, Report HE-43/96/040/A.
32. Hodskinson, A., Ferguson, R.I., (1998), "Numerical Modelling of Separated Flow in River Bends: Model Testing and Experimental investigation of geometric Controls on the Extent of Flow Separation at the Concave Bank", Hydr. Processes, Vol. 12, pp. 1323-1338.
33. Isaacson, E., Stoker, J.J., Troesh, B.A., (1954), "Numerical Solution of Flood Prediction and River Regulation Problems (Ohio-Mississipi Floods)", in Cunge, J.A., Holly, F.M., Verwey, A., (1980).
34. Karki, K.C., Patankar, S.V., (1988), "Calculation Procedure for Viscous Incompressible Flows in Complex Geometries", Num. Heat Treansfert, Vol. 14, pp. 295-307.
35. King, I.P., Norton, W.R., (1978), "Recent Application of RMA's Finite Element Models for Two-Dimensional Hydrodynamics and Water Quality", In Brebbia, Gray and Pinder (1978).
36. King, I.P., Roig, L.C., (1991), "Finite Element Modelling of Flow in Wetlands", Proc. National Conf. Hydr. Eng., ASCE, pp. 286-291.
37. Knight, D.W., Shiono, K., (1996), "River Channel and Floodplain Hydraulics", in "Floodplain Processes", M.G. Anderson, D.E. Walling and P.D. Bates, (eds), pp. 139-181.
38. Koutitas, C., O'Connor, B., (1980), "Modelling Three-Dimensional Wind-Induced Flows", J. Hydr. Div., ASCE, Vol. 106, No. HY11, pp.
39. Kuipers, C.B., Vreugdenhil,C.B., (1973), "Calculation of Two Dimensional Horizontal Flow", Delft Hydraulics Laboratory, The Netherlands, Report S163, Part I.
40. Lane, S.N., Bradbrook, K.F., Richards, K.S., Biron, P.A., Roy, A.G., (1999), "The Application of Computational Fluid dynamics to Natural River Channels: Three-Dimensional versus Two-Dimensional Approaches", Geomorphology, Vol. 29, pp. 1-20.
41. Lavedrine, I., (1997), "Evaluation of 3D Models to River Flood Problems", Report TR26, HR Wallingford, UK.

42. Lavedrine, I., (1996), "Evaluation of 3D Models for River Flood Applications", Report TR 6, HR Wallingford, UK.
43. Lean, G.H., Weare, T.J., (1979), "Modelling Two-Dimensional Circulating Flow", J. Hydr. Div., ASCE, Vol. 105, No. HY1, pp. 17-26.
44. Leendertse, J.J., (1967), "Aspects of Computational Model for Long Period water Wave Propagation", RAND Corporation Report, Santa Monica, California, USA, Report RM 5294-PR.
45. Leendertse, J.J., Alexander, R.C., Liu, S.K., (1973), "A Three Dimensional Model for Estuaries and Coastal Seas: v1 Principles of Computation", RAND Corporation Report, Santa Monica, California, USA, Report RM 1417 OWRR.
46. Lynch, D.R., Gray, W.G., (1980), "Finite Element Simulation of Flow in Deforming Regions", J. Comput. Phys., Vol. 36, pp. 135-153.
47. McGuirk, J.J., Rodi, W., (1978), "A Depth-Averaged Mathematical Model for the Near Field of Side Discharges into Open Channel Flow", J. Fluid Mech., Vol. 86, part 4, pp. 761-781.
48. Markhanov, S.S., Vannakrairojn, S., Vanderperre, E.J., (1999), "2D Numerical Model of Flooding in East Bangkok", J. Hydr. Eng., ASCE, Vol. 125, No. 4, pp.
49. Martin-Valdepeñas, J. M., Jiménez, M.A., Hernández, (2000), "Hydrogen Behaviour in Nuclear Power Plant Containment", CFX Academic Conference, Theale, March 2000, Publication available on www.aeat.com/cfx.
50. Mesehle, E.A., Sotiropoulos, F., (2000) "Three-dimensional Numerical Model for open-Channels with Free-Surface Variations", J. Hydr. Res., Vol. 38, No. 2, pp. 115-121.
51. Molinaro, P., Natale, L., (1994), "Modelling Flood Propagation over Initially Dry Areas", ASCE.
52. Morvan, H., Pender, G., Wright, N., Ervine, D.A., (2000), "Three-Dimensional Modelling of the Flow Mechanisms in Flooded Meandering Channels", Proc. of the Int. Symp. On Flood Defence, Kassel, Germany, pp. D153-D161 www.uni-kassel.de/fb14/wasserbau/symposium2000.
53. Naot, D., Rodi, W., (1982), "Calculation of Secondary Currents in Channel Flow", J. Hydr. Div., ASCE, Vol. 108, No. HY8, pp. 948-968.

54. Nezu, I., Rodi, W., (1985), "Experimental Study on Secondary Currents in Open-Channel Flow", 21st IAHR Congress Proc., Melbourne, Australia, Vol. 2, pp. 19-23.
55. Neary, V.S., Sotiropoulos, F., Odgaard, A.J., (1999), "Three-Dimensional Numerical Model of Lateral Intake Inflows", J. of Hydr. Eng., Vol. 125, No. 2, pp. 126-140.
56. Nguyen, V.T., Nestmann, F., Eisenhauer, N., (2000), "Three Dimensional Computation of River Flow", Proc. 4th Int. Conf. Hydroinformatics, Iowa City, USA (CD-Rom).
57. Nilsson, H.O., Holmberg, S, (2000), "Improved Ventilation and Climate Conditions", CFX Academic Conference, Theale, March 2000, Publication available on www.aeat.com/cfx.
58. Pantankar, S.V., Spalding, D.B., (1972), "A Calculation Procedure for Heat, mass and Momentum Transfer in Three-Dimensional Parabolic Flows", Int. J. Heat Mass Transfer, Vol. 15, pp. 1787-1806.
59. Paquier, A., Farissier, P., (1997), "Assessment of Risks of Flooding by Use of a 2D Model", IAHS Publication, No. 239, pp. 137-143.
60. Platzmann, G.W., (1958), "A Numerical Computation of the Surge of 26 June 1954 on Lake Michigan", Geophysica, Vol. 6, pp. 407-438.
61. Peltier, E., Le Normant, C., Teisson, C., Malcherek, A., Markofsky, M., Zielke, W., Cornelisse, J., Molinaro, P., Corti, S., Greco, G., (1996), "Three Dimensional Numerical Modelling of Cohesive Sediment Transport Processes in Estuarine Environments", Final Report to EC Contract MAS2-CT92-0013, EDF-DER, LNH, Chatou, France, Report HE-42/96/047/A.
62. Preissman, A., (1961), *"Propagation des Intumescences dans les Canaux et Rivières"*, First Congress of the French Association for Computation, Grenoble, France, (In French).
63. Rastogi, A.K., Rodi, W., (1978), "Predictions of Heat and Mass transfer in Open Channels", J. Hydr. Div., Vol. 104, No. HY3, pp. 397-420.
64. Rozovskii, I.L., (1957), "Flows of water in Bends of Open Channel", Academy of Sciences of the Ukrainian SSR, Kiev, Ukraine (USSR).
65. Shimizu, Y., Yamaguchi, H., Itakura, T., (1990), "Three Dimensional Computation of Flow and Bed Deformation", J. Hydr. Eng., ASCE, Vol. 116, No. 9, pp. 1090-1108.

66. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.
67. Sleigh, P.A., Gaskell, P.H., Berzins, M., Wright, N.G., (1998), "An Unstructured Finite-Volume Algorithm for Predicting Flow in Rivers and Estuaries", Comp. and Fluids, Vol. 27, No. 4, pp. 479-508.
68. Sofiadilis, D., Prinos, P., (1999), "Numerical Study of Momentum Exchange in Compound Channel Flow", J. Hydr. Eng., Vol. 125, No. 2, pp. 152-165.
69. Stone, H.L., (1968), "Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations", SIAM Journal on Numerical Analysis, Vol. 5, Issue 3, pp. 530-558.
70. Swindale, N., (1999), "Numerical modelling of River Rehabilitation", PhD Thesis, The University of Nottingham, UK.
71. Tchamen, G.W., Kahawita, R.A., (1998), "Modelling Wetting and Drying Effects over Complex Topography", Hydr. Processes, Vol. 12, pp. 1151-1182.
72. Thomas, T.G., Willimas, J.J.R., (1995a), "Large Eddy Simulation of a Symmetric Trapezoidal Channel at a Reynolds Number of 430,000", J. Hydr. Res., Vol. 33, No. 6, pp. 825-842.
73. Thomas, T.G., Willimas, J.J.R., (1995b), "Large eddy Simulation of Turbulent Flow in an assymetric Compound Open Channel", ", J. Hydr. Res., Vol. 33, No. 1, pp. 27-41.
74. Tominaga, A., Nezu, I., Ezaki, K., Nakagawa, H., (1989), "Three-Dimensional Turbulent Structure in Straight Open Channel Flows", J. Hydr. Res., Vol. 27, No. 1, pp. 149-173.
75. Van Rijn, L.C., (1987), "Mathematical Modelling of Morphological Processes in the Case of Suspended Sediment Transport", Deflt Hydraulic Communication No. 382.
76. Vreugdenhil, C.B., Wijbenga, J.H.A., (1982), "Computation of Flow Patterns in Rivers", J. Hydr. Div., ASCE., Vol. 108, No. HY11, pp. 1296-1310.
77. Wang, S.Y., Alonso, V.V., Brebbia, C.A., Gray, W.G., Pinder, G.F., (1989), "Finite Elements in Water Resources", Proc. 3rd Int. Conf., Mississipi, USA.
78. Wijbenga, J.H.A., (1985), " Determination of Flow Patterns in rivers with Curvilinear Coordinates", Proc. 21st IAHR Congress, Melbourne, Australia, pp. 131.

79. Wu, W., Rodi, W., Wenka, T., (2000), "Three-Dimensional Numerical Modelling of Flow and Sediment Transport in Open Channels", J. Hydr. Eng., Vol. 126, No. 1, pp.4-15.
80. Yen, B.C., (1965), "Characteristics of Subcritical Flow in a Meandering Channel", Institute of Hydraulic Research, University of Iowa, Iowa City, Iowa, USA.
81. Zoppou, C., Roberts, S., (1999), "Catastrophic Collapse of Water Supply Reservoirs in Urban Areas"J. Hydr. Eng., ASCE. Vol. 125, No.7, pp. 686-695.

| Investigator | Flow Problem | Governing Equations Flow | | Boundary Conditions Solid Wall | Free- | Numerical Solution Num. Scheme | Num. | Comments |
|---|---|---|---|---|---|---|---|---|
| **Rastogi and Rodi (1978)** | Straight, rect. channel | RANS (parabolic) | $k$-$\varepsilon$ | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Good comparison with lab data |
| **Leschziner and Rodi (1979)** | Strongly curved channel | RANS (parabolic) | $k$-$\varepsilon$ | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Improvement on pressure term |
| **Naot and Rodi (1982)** | Straight, rect. channel | RANS (parabolic) | ASM | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Development of ASM |
| **Alfrink and van Rijn (1983)** | Flow over a trench | RANS (full) | $k$-$\varepsilon$ | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Sensitivity of k-ε parameters; Little sensitivity of solution to inlet BC |
| **Krishnappan and Lau (1986)** | Straight, rect., compound channel | RANS (parabolic) | ASM | Wall function | Rigid lid (symmetry, except ε) | Hybrid and CDS + SIMPLE | TDMA | Shear stresses and discharges compared with lab. data |
| **Gibson and Rodi (1989)** | Straight, rect. channel | RANS (full) | RSM | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Modified BC for ε at the free surface |
| **Demuren (1993)** | Meandering channel | RANS (full) | $k$-$\varepsilon$ | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Comparison of velocity with lab. data |

(to be continued, next page)

| Investigator | Flow Problem | Governing Equations: Flow | Governing Equations: Turbulence | Boundary Conditions: Solid Wall | Boundary Conditions: Free-Surface | Numerical Solution: Num. Scheme | Numerical Solution: Num. Solver | Comments |
|---|---|---|---|---|---|---|---|---|
| **Cokljat and Younis (1995)** | Straight, rect. and compound channels | RANS (parabolic) | Non-lin. $k$-$\varepsilon$ RSM | Wall function | Rigid lid (symmetry) | Hybrid and CDS + SIMPLE | TDMA | Full RSM Reproduction of 2ndary currents |
| **Thomas and Williams (1995)** | Straight, rect. compound channels | LES with Smagorinski (1963) | | Wall function | Rigid lid (symmetry) | Modified Schumann (1975) | - | LES applied to FCF: high detail |
| **Sinha *et al.* (1998)** | Natural, straight river reach | RANS (full) | $k$-$\varepsilon$ | Two-point wall function | Rigid lid (symmetry) | 2nd order upwind and CDS + SIMPLE | LU (mom. and turb.): ADI (pressure) | Practical river application: detailed evaluation |
| **Sofialidis and Prinos (1999)** | Straight, rect. compound channels | RANS (full) | Low Reynolds non-lin. k-ω | Resolution of viscous sub-layer | Rigid lid (symmetry, redistribution of normal stresses) | QUICK + PISO | TDMA | Comparison of velocity, 2ndary currents and turb. data |
| **Meselhe and Sotiropoulos (1999)** | Meandering channel | RANS (full) | $k$-$\varepsilon$ | Wall function | Deformable lid (pressure) | First order upwind | ADI | Little contribution of deform lid |
| **Wu, Rodi and Wenka (2000)** | Sediment transport, flow in a bend | RANS (full) | $k$-$\varepsilon$ | Two-point wall function | Deformable lid (2D calculation) | Hybrid and CDS + SIMPLE | Stone method (1968) | Sediment transport model (van Rijn, 1987) |

Underlined Text: Main contribution or focus of the paper with regard to open-channel application of CFD

**TABLE 3.1 – Synthetic Review of Open-Channel Flow Publications using CFD**

# Chapter 4: Grids, Boundary Conditions, Solution Techniques and other Numerical Issues

# Chapter 4: Grids, Boundary Conditions, Solution Techniques and other Numerical Issues

*"The area of numerical grid generation is relatively young in practice, although its roots in mathematics are old. This somewhat eclectic area involves the engineer's feel for physical behaviour, the mathematician's understanding of functional behaviour, and a lot of imagination, with perhaps a little help from Urania".*

(Thompson *et al.*, 1985)

*"Numerical precision is the very soul of science".*

(Sir D'Arcy Wentworth Thompson, 1917; in Anderson, 1995)

## 4.1 INTRODUCTION

This chapter is concerned with the presentation of the main numerical issues that are relevant to the open-channel flow simulation work undertaken by the author. Because of the different nature of the techniques implemented in CFX and TELEMAC, two separate sections are necessary to present each code's specific approach. Both address the same issues, namely the design of the mesh, the choice of the boundary conditions, the discretization scheme and solver, as well as the convergence criterion. Because CFX and TELEMAC also use different discretization methods to solve the Navier-Stokes equations, these "code-specific" sections are preceded by a general overview about finite volume and finite element methods.

For CFX, it was felt necessary to emphasise these topics because CFX is a fully-3D multi-purpose Computational Fluid Dynamics (CFD) code. It is therefore very general and demands a precise numerical set up to be successfully applied. The complexity of the set up is mostly related to the three-dimensional nature of the code, which requires special care regarding the problem closure in the vertical dimension. The treatment of the free surface and the representation of the boundary layer at the walls are difficult questions that require to be addressed

TELEMAC is a quasi-3D code developed on the base of the generalised two-dimensional St Venant equations. These assume a vertical hydrostatic pressure distribution. It is also solely dedicated to fluvial, estuarine and coastal hydraulic problems. The third dimension is in fact only partly resolved in TELEMAC. The water height is first calculated on a two-dimensional mesh, which is then replicated on the vertical in 'n' bottom-fitted layers. The main horizontal velocity field is calculated as a layer-averaged problem, and the vertical velocities calculated by closure on the continuity equation. This approach avoids the difficulties of the representation of the free surface in fully-3D codes. However, vertical velocities and pressure are only approximately calculated.

## 4.2 FINITE VOLUME METHOD VERSUS FINITE ELEMENT

### 4.2.1 Finite Volume (CFX)

#### 4.2.1.1 Basic Principles

The finite volume method proceeds by integrating the conservation laws, expressed in the Navier-Stokes equations for a control volume, over the entire domain, prior to the discretization phase. This ensures the exact conservation of the fluid physical properties in a control volume (CV) and yields a simple, physically-based, formulation of the form:

$$\begin{bmatrix}\text{Rate of change of} \\ \text{a property } \phi \text{ in the} \\ \text{CV with respect to time}\end{bmatrix} = \begin{bmatrix}\text{Net flux of } \phi \text{ due to} \\ \text{convection into the CV}\end{bmatrix} + \begin{bmatrix}\text{Net flux of } \phi \text{ due to} \\ \text{diffusion into the CV}\end{bmatrix} + \begin{bmatrix}\text{Net rate of creation} \\ \text{of } \phi \text{ inside the CV}\end{bmatrix}$$

(Versteeg and Malalasekera, 1995)

The space domain is subdivided into a set of non-overlapping cells, on which the fluid conservation properties are applied. This enables discrete fluid flow variables to be determined at cell nodes. As with the finite difference method the representation of the solution remains solely nodal in the finite volume technique, and no interpolation structure is intrinsically built in the grid.

In the finite volume approach, the Navier-Stokes equations are discretized on the grid, but for control volumes that do not need to correspond to the grid cells. Grid cells are used to represent the geometry and carry the problem variables. Control volumes are regions of the discretized domain on which the conservation laws are applied. They can be chosen independently to suit the problem physics. Control volumes make use of the grid cell structure to calculate the different components of the flow property, but they can be made of fractions of cells or several cells. This reinforces the method's popularity because it offers the possibility to solve the problem on the most appropriate control volume and meet the geometry requirements on a different structure simultaneously. Grid cells can have varied shapes to fit the problem topography. The location of the nodes, where the problem data and variables are stored, on the other hand is usually chosen to accommodate the solution algorithm. They can be located at the vertices, at the cell centre or in other locations. In addition, for a given grid cell, some nodes can be used to represent one particular flow property, while the remaining nodes are used for the others. A classical use of this feature is made when designing a staggered grid (Harlow and

Welch, 1965), Fig. 4.1(a). This standard type of grid stores the scalar flow properties at the grid vertices (e.g. the pressure or temperature) and the fields on a second grid (staggered grid) centered on the first grid vertices. The staggered grid represents the frame on which the control volumes are chosen, and forms a second grid. This technique is very popular because it avoids the classical "checker-board" pressure problem (Versteeg and Malalasekera, 1995). However this is not the only approach, and indeed, a non-staggered grid can be implemented safely (e.g. in CFX) by making use of the Rhie-Chow algorithm (1983) to alleviate the pressure problem underlined above. In the non-staggered grid approach all the variables are stored on the same grid, at a central node within each control volume, Fig. 4.1(b). This simplifies greatly the grid structure, although the introduction of an extra numerical term introduces a slight inaccuracy. The fluxes at the faces of the control volume are calculated by interpolation of the nodal values.

Once this discretization work is complete, and the Navier-Stokes equations have been integrated over each of the control volumes, simple interpolation techniques are used to discretize them locally on the control volumes.

4.2.1.2 Finite Volume Formulation

The intuitive aspects of the finite volume method, which derive from Taylor series expansion, are exposed in the following formulation. Recalling that the Navier-Stokes equations becomes after a formal integration:

$$I = \int_{CV} \left( \rho \cdot \frac{\partial \vec{U}}{\partial t} + \rho \vec{U} \cdot \nabla \cdot \vec{U} + \nabla \cdot p - \rho \cdot \nabla \cdot \left( \nu \nabla \vec{U} \right) - \vec{B} \right) dV = 0$$

$$I = \underbrace{\int_{CV} \rho \cdot \frac{\partial \vec{U}}{\partial t} dV}_{\substack{\text{Rate of Change} \\ \text{with respect to time}}} + \underbrace{\int_{A} \rho \vec{U}\vec{U} \cdot \vec{n} dS}_{\substack{\text{Fluxes through} \\ \text{the CV faces A}}} + \underbrace{\int_{CV} \nabla \cdot p dV}_{\text{Pressure}} - \underbrace{\int_{A} \rho \nu (\nabla \vec{U}) dS}_{\text{Diffusion through A}} - \underbrace{\int_{CV} \vec{B} dV}_{\substack{\text{Source} \\ \text{Terms}}} = 0 \tag{4.1}$$

Considering equation (4.1) in two dimensions on a grid made of hexahedral elements, the advection term can be discretized within the control volume along the $x$-direction ($U$) as:

$$
\begin{aligned}
\int_A \rho\vec{U}\vec{U}\cdot\vec{n}dS &= [(\rho UU\cdot A)_e - (\rho UU\cdot A)_w] + [(\rho VU\cdot A)_n - (\rho VU\cdot A)_s] \\
&= [(F_u\cdot U)_e - (F_u\cdot U)_w] + [(F_v\cdot U)_n - (F_v\cdot U)_s] \\
&= \left[F_{u_e}(\theta_u\cdot U_E + (1-\theta_u)\cdot U_P) - F_{u_w}(\theta_u\cdot U_P + (1-\theta_u)\cdot U_W)\right] \\
&\quad + \left[F_{v_n}(\theta_v\cdot U_N + (1-\theta_v)\cdot U_P) - F_{v_s}(\theta_v\cdot U_P + (1-\theta_v)\cdot U_S)\right]
\end{aligned} \tag{4.2a}
$$

The diffusion term as:

$$
\begin{aligned}
\int_A \rho\nu\cdot(\nabla\vec{U})dS &= \left[\left(\nu\cdot\frac{\delta U}{\delta x}\cdot A\right)_e - \left(\nu\cdot\frac{\delta U}{\delta x}\cdot A\right)_w\right] + \left[\left(\nu\cdot\frac{\delta U}{\delta y}\cdot A\right)_n - \left(\nu\cdot\frac{\delta U}{\delta y}\cdot A\right)_s\right] \\
&= \left[\left(\frac{\nu\cdot A}{\delta x}\cdot\delta U\right)_e - \left(\frac{\nu\cdot A}{\delta x}\cdot\delta U\right)_w\right] + \left[\left(\frac{\nu\cdot A}{\delta y}\cdot\delta U\right)_n - \left(\frac{\nu\cdot A}{\delta y}\cdot\delta U\right)_s\right] \\
&= [D_e(U_E - U_P) - D_w(U_P - U_W)] + [D_n(U_N - U_P) - D_s(U_P - U_S)]
\end{aligned} \tag{4.2b}
$$

Finally the source term as:

$$
\int_V \vec{B}dV = \bar{S}\cdot\Delta V_u \tag{4.2c}
$$

The above calculation is conducted in an identical manner in the $y$-direction ($V$). The indices $W$, $N$, $E$ and $S$ to refer to the nodes surrounding $P$ in two dimensions and, $w$, $n$, $e$ and $s$ to the control volume faces located between $W$ and $P$, $N$ and $P$, $E$ and $P$, and $S$ and $P$ respectively (Fig. 4.1). In the following, upper case letters will refer to nodal values or their positions on the grid, whereas lower case letters will indicate values and positions at control volume faces. In two dimensions, both types of letters will be combined to indicate the location of the variables in the plane grid $(x,y)$ or $(I,J)$. $F$ represents the flux through a uniform control volume face $A$ calculated from the previous time-step in a decoupled manner. $\theta$ is an implication parameter. $D$ is called the diffusion conductance and the ratio of $F$ over $D$ is called the Péclet number, $Pe$. This is indicative of the relative strength of advection compared to diffusion (see 4.3.2): $Pe$ = 0 signifies a pure diffusion case, whereas a high $Pe$ indicates that the flow is advection-dominated. $\bar{S}$ is the average source term over the control volume. $\delta x$ and $\delta y$ represent the grid spacing, and $\delta U$ and $\delta V$

the finite differences for the velocity terms, which are used to approximate the differential terms.

Using (4.2) in a simplified version of (4.1) leads to:

$$\begin{aligned}
&\left(F_{u_e}\cdot(1-\theta_u)-F_{u_w}\cdot\theta_u+F_{v_n}\cdot(1-\theta_v)-F_{v_s}\cdot\theta_v+(D_e+D_w+D_n+D_s)\right)\cdot U_P\\
&\quad-\left(F_{u_w}\cdot(1-\theta_u)+D_w\right)\cdot U_W-\left(-F_{u_e}\cdot\theta_u+D_e\right)\cdot U_E\\
&\quad-\left(-F_{v_n}\cdot\theta_v+D_n\right)\cdot U_N-\left(F_{v_s}\cdot(1-\theta_v)+D_s\right)\cdot U_S\\
&\quad=\overline{S}\cdot\Delta V_u
\end{aligned}\tag{4.3}$$

This can be generalized in two dimensions under the form:

$$\begin{aligned}
&-a_{i-1,J}\cdot U_{i-1,J}-a_{i,J-1}\cdot U_{i,J-1}+a_{i,J}\cdot U_{i,J}-a_{i,J+1}\cdot U_{i,J+1}-a_{i+1,J}\cdot U_{i+1,J}=b_{i,J}\\
&-a_{I,j-1}\cdot V_{I,j-1}-a_{I-1,j}\cdot V_{I-1,j}+a_{I,j}\cdot V_{I,j}-a_{I+1,j}\cdot V_{I+1,j}-a_{I,j+1}\cdot V_{I,j+1}=b_{I,j}
\end{aligned}\tag{4.4}$$

In which the $a_{i,J}$ and $a_{I,j}$ represent the appropriate combination of the fluxes and diffusion terms from (4.3). The diagonal-band structure of the system matrices appears clearly in (4.4) for simple problems on well-ordered grids. This is a numerical attribute that should be exploited when solving the system of equations obtained with the above discretization. In cases where the numbering of the nodes is not well ordered on the grid, or the discretization technique is more complex (e.g. QUICK), (4.4) can yield a sparse matrix that requires a sophisticated numerical technique to be solved.

The pressure terms need to be added in (4.4). These terms can be treated as:

$$\begin{aligned}
\frac{p_P-p_W}{\Delta x_u}\cdot\Delta V_u&=(p_P-p_W)\cdot A\\
\frac{p_P-p_S}{\Delta y_v}\cdot\Delta V_v&=(p_P-p_S)\cdot A
\end{aligned}\tag{4.5}$$

Consequently, (4.4) becomes:

$$\begin{aligned}
&-a_{i-1,J}\cdot U_{i-1,J}-a_{i,J-1}\cdot U_{i,J-1}+a_{i,J}\cdot U_{i,J}-a_{i,J+1}\cdot U_{i,J+1}-a_{i+1,J}\cdot U_{i+1,J}+\left(p_{I,J}-p_{I-1,J}\right)\cdot A_{i,J}=b_{i,J}\\
&-a_{I,j-1}\cdot V_{I,j-1}-a_{I-1,j}\cdot V_{I-1,j}+a_{I,j}\cdot V_{I,j}-a_{I+1,j}\cdot V_{I+1,j}-a_{I,j+1}\cdot V_{I,j+1}+\left(p_{I,J}-p_{I,J-1}\right)\cdot A_{I,J}=b_{I,j}
\end{aligned}$$

Which is usually written,

$$\begin{aligned} a_{i,J} \cdot u_{i,J} &= \sum_{\substack{\text{neighbouring}\\ \text{nodes nb}}} a_{nb} \cdot u_{nb} + \left(p_{I-1,J} - p_{I,J}\right) \cdot A_{i,J} + b_{i,J} \\ a_{I,j} \cdot u_{I,j} &= \sum_{\substack{\text{neighbouring}\\ \text{nodes nb}}} a_{nb} \cdot u_{nb} + +\left(p_{I,J-1} - p_{I,J}\right) \cdot A_{I,J} + b_{I,J} \end{aligned} \tag{4.6}$$

A pressure-coupling technique is needed to work out a relationship between pressure and velocity variables (see 4.3.4). This brief introduction already illustrates the simplicity of the finite volume formulation, which appeals to the CFD programmers.

### 4.2.2 Finite Element (TELEMAC)

#### 4.2.2.1 Basic Principles

The finite element formulation relies on a very solid mathematical base. It is probably the most rigorous way of discretizing the Navier-Stokes equations, notably the diffusion part (Idelsohn and Oñate, 1994).

The geometrical space is discretized into a series of elements within which the Navier-Stokes equations are valid. However, the finite element construction of the geometry elements implies both a geometrical and function-space discretization. In the finite element method the discretization of the partial differential equations possesses a true spatial dimension, since it can be continuously interpolated inside each element thanks to polynomial functions that relate the continuous variables in space to nodal values. These polynomials assume that the solution of the variables has a particular prescribed form: "*the solution has to belong to a function space*" (Dick, 1996). For example if the polynomials are parabolic, it is assumed that the variables will vary accordingly inside an element. They represent a base on which an approximate solution to the Navier-Stokes equations is decomposed as a function of the nodal values. Inside each element, a variable $\phi$ is written:

$$\phi = \sum_{I}^{\substack{\text{Number of}\\ \text{Element Nodes}}} N_I \cdot \phi_I \tag{4.7}$$

Where the $N_l$ represent the polynomial base, which are called the shape functions, and $\phi_l$ the values of the variable sought at node $l$. (For a pictorial illustration of the shape function, see Fig. 4.2)

Then,

$$\frac{\partial \phi}{\partial x} = \sum_{l}^{\substack{\text{Number of} \\ \text{Element Nodes}}} \frac{\partial N_l}{\partial x} \cdot \phi_l \tag{4.8}$$

This decomposition of the continuous functions entails some conditions regarding the convergence of the model towards the correct numerical solution. Convergence is the property of a numerical method to produce a solution that approaches the exact solution as the grid spacing is reduced to zero (Versteeg and Malalasekera, 1995). This condition is called the completeness or consistency criterion and is common to all discretization techniques. A particular requirement due to (4.8) however, is that if the governing equation is of order n then the variable and its derivative must be continuous across the boundary to the order of at least n-1. This is the compatibility criterion (Lee and Froelich, 1986; Finnie, 1994).

There are three different approaches regarding the application of the finite element method: direct, variational or weighted-residual. Unfortunately, the first two methods cannot be easily applied to the Navier-Stokes equations, since a finite element formulation cannot be derived from an energy consideration. Such a formulation needs to be written directly. As a result, it is the residual method that is applicable to most fluid mechanics problems.

Firstly, an approximation to the solution is written *per element* (4.7) as a function of the nodal variables for which a solution is required. These approximations meet the consistency requirement. Then the compatibility requirement is met by adjusting the individual approximations to be continuous across *the entire domain*. This means that the discretized element functions are adjusted to be continuous across their boundaries. The residual of the Navier-Stokes equations discretized in this fashion is then minimised by adjusting of the nodal variable values. An illustration of this process is provided on Fig. 4.3 for a one-dimensional problem, assuming first order shape functions.

4.2.2.2 Finite Element Formulation

Inside the weighted-residual methods, there exist three formulations to achieve minimisation: the collocation, least squares or Galerkin's formulations. Among the three, Galerkin has been the favoured formulation in free-surface fluid flow simulation, in particular for the treatment of the diffusion terms.

This formulation is illustrated here using a simplified form of the Navier-Stokes equations in which the index $i$ refers to the spatial directions $(x,y,z)$ direction, $l$ and $m$ are two indices representing the node numbers, and $k$ is the number of discrete elements. The superscript $n$ indicates the present time step, and $\Delta t$ the time increment (TELEMAC models unsteady flows). $\vec{U}_l$ is the velocity vector at node $l$, $\vec{U}_c$ is the convection velocity which is assumed to be known a priori, in order to simplify the non-linearity. Conservation and momentum equations can therefore be expressed on a finite element $k$ (sum over element $k$), as:

$$\sum_l \vec{U}_l \cdot \frac{\partial N_l}{\partial x_i} = 0 \tag{4.9}$$

$$\sum_l \frac{\vec{U}_l^{n+1} - \vec{U}_l^n}{\Delta t} \cdot N_l + \left( \sum_l \vec{U}_c \cdot \vec{U}_l \cdot \frac{\partial N_l}{\partial x_j} \right) - \left( \sum_l \vec{U}_l \cdot \nabla \cdot \left[ \nu \cdot \nabla (N_l) \right] \right) = \vec{B} \tag{4.10}$$

These relationships need to be extended to the entire domain. One solution to generalize the above expression is to integrate this differential formulation over the entire domain to formulate what is known as the weak statement of the differential equations problem. After some manipulations (4.9) and (4.10) become (sum over all the elements):

$$\sum_l \vec{U}_l \cdot \left( \underbrace{\int_\Omega W_m \cdot \frac{\partial N_l}{\partial x_i} d\Omega}_{\text{"Gradient Matrix"}} \right) = 0 \tag{4.11}$$

$$\sum_l \frac{\vec{U}_l^{n+1}-\vec{U}_l^n}{\Delta t}\left(\underbrace{\int_\Omega W_m \cdot N_l d\Omega}_{\text{Mass Matrix}}\right)+\sum_l \vec{U}_l\left(\underbrace{\int_\Omega \vec{U}_c \cdot \frac{\partial N_l}{\partial x_i} \cdot W_m d\Omega}_{\text{Advection Matrix}}\right)-\sum_l \vec{U}_l\left(\underbrace{\int_\Omega \nabla \cdot [\nu \cdot \nabla(N_l)] d\Omega}_{\text{"Gradient-Divergence Matrix"}}\right)=\int_\Omega \vec{B} d\Omega \tag{4.12}$$

($\Omega$) represents the solution domain, and $W$ the weights, expressed as a function of the shape terms $N$ in the Galerkin-based methods. Once the weight and shape functions have been calculated in (4.12) for example, the problem clearly resembles that formulated in (4.6). Here the integral forms replace the $a_{i,j}$ terms in (4.6) and are simply a more rigorous calculation of the advection, diffusion and mass terms.

### 4.2.3 Discussion

Both finite element and finite volume techniques enable the discretization of the Navier-Stokes equations. It can be said that they are numerically equivalent, e.g. if the weight functions are chosen equal to a constant (sub-domain collocation) or when the finite volume discretization adopts a finite-element-like type of formulation (Dick, 1996). This is illustrated by the similarities between (4.6) and (4.12). Yet the two methods are also very different. One is conceptually closer to basic physical processes and is more easily programmed in an iterative manner, whereas the other one necessitates a much stricter mathematical formulation. The interested reader is referred to the excellent comparative paper of Idelsohn and Oñate (1994).

The finite volume method first defines the basic physical principles involved in the problem, i.e. the conservation principle, in an integral form. Since it is at the core of the method, this principle will be satisfied at all times. A simple derivation of the problem equations is obtained by application of finite differences, and the resulting numerical problem is usually simple to solve. Instead of trying to tackle the mathematical problem directly the latter is simplified and an iterative trial-and-error process is implemented until an acceptable solution is reached. This is easier to implement computationally and can be efficiently managed, which makes the method particularly suited for large and/or complex problems especially on structured grids. It is the most commonly applied method in CFD codes because it is very efficient to treat advection dominated equations, and has been extended to unstructured grids (CFX 5).

In the author's opinion the finite element technique presents a more rigorous treatment of the mathematical problem. The partial differential equations are discretized with rigour at an infinitesimal level assuming that their solution belongs to a hypothetical function space, before being integrated. However the method then aims to minimise the residuals in a global sense. The physics and the discrete nature of the problem are not considered locally and a drawback of the method is that it has major difficulties meeting the flow conservation requirement unless the mesh is very fine. Additionally, imposing a known depth as a boundary condition is incompatible with the finite element formulation of the conservation equation. Some of these issues are detailed in TELEMAC-2D Technical Notes (Hervouet and van Haren, 1995)

## 4.3 CFX NUMERICAL ISSUES AND MATHEMATICAL ASSUMPTIONS

### 4.3.1 Spatial Discretization: Definition of the Geometry and the Mesh

The version of CFX (CFX 4.2, 1997) that was used for the following tests relies on a structured body-fitted grid. This is justified by the fact that CFX was originally developed to model fluid problems where the geometry is relatively simple. An unstructured version of the code (CFX 5) is also marketed, but was not as complete as CFX 4 at the start of this research project.

#### 4.3.1.1 The Geometry

There are two major issues regarding the construction of the geometry:

(i) the construction of the geometry components (lines, surfaces and solids)

(ii) the evaluation of the topological properties of the geometry elements with respect to meshing requirements (in particular smoothness of the lines and cell aspect ratio).

Prior to constructing the mesh, it is necessary to define the geometry, which is the volume that will contain the fluid and its external boundaries (inlet, bed, free surface and outlet). These volumes, or blocks, are made of assembled surfaces, some of which constitute the domain boundary where "boundary conditions" are applied. The surfaces are usually made of interconnected curves, defined as segment lines or splines (quadratic or cubic). These curves pass through a series of survey points that describe the geometry. For river geometry this definition of the geometry is awkward and arises from the fact that CFX is designed for downloading problem geometry from Computer-Aided Design (CAD) software.

For a very simple domain, it is possible to define the geometry accurately in a monolithic fashion. However, in most cases this is not possible, and the use of multiple blocks to represent simple parts of a complex geometry is recommended. These blocks are interconnected at common surfaces to form the complete body. These surfaces are treated as "internal boundary conditions" by the solver. The use of the multi-block technique permits the description of complex shapes. Therefore modelling most man-made structures (ducts, airfoils) is relatively straightforward from a geometrical point of view. A direct consequence is that there is little difficulty implementing the mesh on such simple volumes. This is not always the case however. As complexity increases the topological properties of the geometry often have to be accounted for in order to build a good grid.

The term topology refers to the shape properties of the constructed blocks and their connections. It is an important issue to consider in the early stages of the geometry construction, as it later impacts on the meshability of the geometry and the quality of the numerical solution. For example, if the geometry lines are smooth and regular, or if different blocks are constructed to make the geometry smooth and regular (Fig. 4.4), the construction of the mesh cells is facilitated and their aspect ratio properties are satisfactory. The sigma-transformation from this grid into a regular grid where the finite volume calculation is conducted is then simplified. This creates a simple and reliable frame on which the Navier-Stokes equations can be discretized. Another example concerns the implementation of boundary conditions (either internal or external). If the connection between two blocks is complex and irregular (or if any other open or solid boundary is), this can cause problems in the implementation of boundary conditions, and can create inaccurate mass flow calculation for example. This generally affects convergence and stability, especially in turbulent flows.

The above examples have illustrated some classical difficulties that can be met while constructing the geometry. With natural river channels, irregularity is already present in the geometry prior to its discretization, which creates further cause for concern.

In general, river surveys will be carried out so as to produce a discrete measure of the river main channel geometry at cross-sections, located every 50-100 m typically. In the present work, detailed measurements of the bank lines were also taken to build the skeleton of the three-dimensional blocks. Natural geometry is continuous. Consequently one should construct as many blocks as possible to describe the geometry:

$$\text{"True" Geometry} = \lim_{\infty}\left(\text{Number of Blocks}\right) \qquad (4.13)$$

In reality, one is obviously limited by the resolution of the data collected on the site:

$$\text{"Optimum" Geometry} = \lim_{\substack{\text{Topographical} \\ \text{Data Density}}}\begin{pmatrix}\text{Number of Solids based} \\ \text{on Cross - Sectional Survey}\end{pmatrix} \qquad (4.14)$$

This is symbolised on Fig. 4.5. It could be argued that a multitude of blocks is detrimental to the CPU performance, as each block is solved independently during the solution process. In reality this is not an issue in CFX as the mesh multi-block structure can be made different from that of the geometry by using a mesh post processor called Meshimport. In the case of the River Severn a simple 26-block structure for the geometry (Fig 4.6) was simplified to a computationally equivalent 3-block structure after the mesh generation.

To the author, the best approach is to build blocks between the main cross-sections. Each cross-section can be constructed using a composite edge made of several lines between the data points. Splines can also be used to enhance smoothness and smooth-out small dents along the line, but this approach can be impractical for parts of the banks in particular, in "shallow" regions as illustrated on Fig. 4.7 (over-shoot). On the other hand splines are recommended in the planview to smooth the bank lines (Fig. 4.8). This has been found to enhance the calculation of pressure by limiting its oscillations (Sinha *et al.*, 1996). Such measure seemed essential in order to obtain a fully converged solution on the pressure term in some cases. Attention should also be paid to ensure a smooth connection of the bank lines at the cross-section interface (Fig. 4.9).

Once the main cross-section and bank lines are constructed, it is relatively straightforward to build the bed bottom surface, by fitting a surface to these edges and by projecting it onto intermediate cross-section lines (Fig. 4.10). From the knowledge of the local position of the free surface it is possible to position a surface (a plane in the simplest cases) "above" the bed surface to represent it. The block sides can then be constructed by projection of the bed surface on the free surface.

The optimum design of an overbank flow geometry is obtained by constructing two separate block structures on top of each other, one for the main channel (inbank), the other for the flooded flood plain. The main channel geometry is chosen as the main constraint in the design of the block shapes, as it constitutes the strongest topographical feature (Fig. 4.6).

4.3.1.2 Mesh Construction

On each curve of the geometry, reference nodes, or "seeds", are positioned to form the frame of a surface mesh. One can chose to implement a regular seed frame, or, in order to enhance the resolution on the limits of the domain, implement biased distributions as described hereafter, Fig. 4.11. Two options have to be considered in order to implement such a refinement procedure on a given curve terminated by two ending points:

(i) the creation of a finer mesh close to one end only, Fig. 4.11(a);

(ii) the creation of a finer mesh at both ends, Fig. 4.11(b).

The mathematical description of the series used to implement the mesh refinement in CFX is derived hereafter.

If a given dimension is split using a one-directional bias, Fig. 4.11(a), e.g. from the bed towards the surface, with a growth coefficient $r$ so that the larger element is $r$-times larger than the first one ($h_1$), and for $n$ layers, each layer has a height $h_j$:

$$h_j = \alpha \cdot h_{j-1} = (\alpha)^{j-1} \cdot h_1$$

and, (4.15)

$$h_1 = \frac{h}{\left(\dfrac{1-\alpha^n}{1-\alpha}\right)} \quad \text{with } \alpha^{n-1} = r$$

With $h$, the total segment length and $\alpha$ the growth coefficient between two adjacent cells.

This results from the calculation of a simple series and allows for a controlled distribution of the increase between two adjacent layers.

Another approach is to consider that both edges, on a given dimension, require a finer resolution. For example, this can be required over the width of a channel to refine the regions close to both walls or banks. In this case a bi-directional bias is required, Fig. 4.11(b). Two algorithms are necessary as follow:

If $n$ is even,

$$\forall j \le n/2,\ h_j = \alpha \cdot h_{j-1} = (\alpha)^{j-1} \cdot h_1$$
$$h_{n-j+1} = h_j \qquad (4.16)$$

and,

$$h_1 = \frac{h/2}{\left(\dfrac{1-\alpha^{n/2}}{1-\alpha}\right)} \quad \text{with } \alpha^{n/2-1} = \alpha^{(n-2)/2} = r$$

If $n$ is odd,

$$\forall j \le (n-1)/2,\ h_j = \alpha \cdot h_{j-1} = (\alpha)^{j-1} \cdot h_1$$
$$h_{n-j+1} = h_j \qquad (4.17)$$

and,

$$h_1 = \frac{h}{\left(2 \cdot \dfrac{1-\alpha^{(n-1)/2}}{1-\alpha} + \alpha^{(n-1)/2}\right)} \quad \text{with } \alpha^{(n-1)/2} = r$$

The use of the above refinement techniques is important in modelling open-channel flows, since it enables improved simulations in areas of important gradients, such as the walls, without having to design a uniformly fine grid. Additionally, the use of geometric series such as those implemented in CFX enable the control of the size increment between two adjacent cells, avoiding inaccurate trial and error procedures.

General guidelines regarding the creation of such grids are:

(i) There should be no abrupt changes in cell aspect ratio between adjacent cells (a ratio inferior than 1.2 is recommended between adjacent cells for CFX, Wright, 2001). This implies that in general $r$ should remain small when the number of elements, $n$, is small over a given edge length; however, when $n$ increases, so should $r$, so that both the resolution at the walls and the overall domain resolution improve;

(ii) There should be no angle of less than 30° between grid lines (Badbrook *et al.*, 2000); this is to avoid diamond shape cells, which are source of numerical instabilities;

(iii) One should refine the grid in regions of strong gradients of the variables, and therefore adapt the mesh to the solution (Sinha *et al.*, 1996).

4.3.1.3 Mesh Independence

The consistency principle dictates that the discrete difference equations should become exact as the grid spacing tends to zero, irrespectively of the numerical scheme used (see 4.2.2). However, one can only use a finite number of elements to carry out a numerical calculation, and consequently some numerical error is going to result. The truncation error for example is usually proportional to a power of the grid spacing. This remark implicitly underlines the fact that spatial and numerical dicretizations are closely related.

One should therefore aim to minimize such numerical error during the design of the numerical model by testing the impact of different levels of spatial discretization on the numerical solution for a given numerical scheme. Richardson interpolation can be conducted on two consecutive grids to assess the level of numerical error between the two. This is however quite time-consuming, and here a more qualitative method is used: Solutions obtained with different grid resolutions are plotted on a same figure and compared. Once the solutions from the different grids appear to converge towards an asymptotic group of values, it is concluded that a "mesh-independent" solution has been reached. This comparison is done once the calculated solutions have all converged satisfactorily and their residuals have reached the required reduction value (see 4.3.6).

Mesh independence is a key issue in numerical modelling. It is also strongly related to the properties of the numerical scheme and the grid. More details can be found in a special issue on verification and validation for CFD by the American Institute for Aerospace and Aeronautics (AIAA, 1998). However in complex situations such as those encountered in rivers, the scale of the geometry and the level of uncertainty in several aspects of the model might affect the significance of such test in comparison with more controlled tests conducted in the field of aeronautics for example. In the Flood Channel Facility (FCF) model mesh independence will be scrupulously evaluated, however the author found it less obvious in the case of the Rivers Severn and Ribble. This will be highlighted in Chapter 6.

### 4.3.2 Numerical Discretization

#### 4.3.2.1 Properties of Discretization Schemes

There are three physical criteria that the numerical schemes must meet:

(i) Conservativeness, the fluxes through a cell are represented in a consistent manner and respect the local mass balance.

(ii) Boundedness, which implies that, in the absence of sources the internal nodal values, a given property should be bounded by its boundary values. This is expressed in a more general fashion by Scarborough (in Versteeg and Malalasekera, 1995) as a boundedness condition on the terms in the equation matrix.

(iii) Transportiveness (Roache, 1976), which is related to the ability of the scheme to determine the flow direction and therefore which surrounding nodes are going to influence the calculation of a given variable at one point. Transportiveness is usually evaluated in terms of the local Péclet number, equal to the ratio of the relative strength of convection and diffusion ( $Pe = [\rho U]/[\nu / \delta x]$ where $\delta x$ is the local grid spacing).

Fourth, fifth and sixth numerical criteria are those of:

(iv) Consistency, i.e. the discretization should become exact as the grid spacing tends to zero.

(v) Numerical accuracy, more specifically related to the discretization error, defined as the difference between the exact solution of the partial differential equations and the exact solution of the algebraic system of equations obtained by discretizing these equations (Ferziger and Peric, 1996).

(vi) Stability, i.e. that the numerical scheme does not generate or magnify errors during calculation

The equations presented in Chapter 2 possess two essential transport properties, those of combined convection and diffusion. Some schemes for example will treat more adequately convection-dominated flows, others diffusive flows. So the choice of a scheme requires serious consideration to ensure that it is suited to the problem that is being considered.

If most numerical schemes are conservative and consistent by construction, boundedness is often an issue, notably regarding the value of the local Péclet number that needs restricting (Versteeg and Malalasekera, 1995). This is quite serious as unbounded schemes may lead to unphysical solutions, as transportiveness, numerical accuracy and ultimately the stability of the iterative solution are affected. Such schemes have to be considered with care, as they will require the grid to be adequately constructed for a given flow to be modelled correctly. This is also the case for any scheme with conditional transportiveness.

Numerical accuracy is a central issue to most CFD problems. If strong gradients of the main variables exist in the solution, first order schemes might smear their profiles very badly, and, the lower the accuracy in terms of the Taylor series truncation error, the more inaccurate irregular profiles will be. This is particularly true for flows with high Reynolds numbers. On the other hand first order schemes are in general more stable, cheaper and easier to run as each nodal solution only involves the immediate-neighbouring nodes. The user has to be aware that a compromise has to be met between accuracy and efficiency.

4.3.2.2 Choice of Discretization Schemes

Several differencing schemes can be used to discretize the convection term in the Navier-Stokes equations: upwind, hybrid, Central Difference Scheme (CDS), QUICK, CCCT amongst others. All of them and some others are available in CFX, and can be selected from the outset by the user.

As in several other commercial codes, the hybrid scheme is the default in CFX, because it produces bounded solutions at all Péclet numbers and is stable. However, it is diffusive and can be rather inaccurate for local Péclet number greater than 2 when it becomes equivalent to a first order upwind scheme. In the hybrid scheme the stability characteristic (of upwind) is gained at the expense of accuracy.

Although more accurate than hybrid and upwind the CDS is in general not suited for general fluid dynamics as it lacks the essential property of transportiveness. It also has limited boundedness properties where the cell Péclet number is greater than or equal to 2. It is therefore particularly inadequate to represent the convection terms. However, it is suitable for the other terms of the Navier-Stokes equations and is used for this purpose in CFX.

QUICK offers third order accuracy since it relies on a quadratic interpolation function. During the course of the present work it has been found to improve numerical results previously obtained with hybrid in the Flood Channel Facility experiment (Chapter 5). It is also non-diffusive. Yet boundedness problems in turbulent problems are a source of concern, especially in region of high gradients where non-physical values can be calculated. The work done at Leeds University by Gaskell and Lau (1988) has produced the CCCT scheme, also more correctly referred to as the QUICK scheme with a SMART limiter function. It is a discretization method based on QUICK, but which is also bounded. It is third order accurate and can deal with sharp field gradients in a physically realistic manner.

In the following hybrid is therefore used whenever suited to improve time-efficiency, and the CCCT scheme is used to enhance the solution obtained with hybrid when the latter scheme generates too much spurious diffusion.

### 4.3.3 Boundary Conditions

Boundary conditions need to be implemented to define the behaviour of the variables on the edges of the model. This is an essential part of the mathematical solution of the Navier-Stokes equations and needs to be properly implemented. Four regions have to be considered: inlet, outlet, walls or bed and banks, and the free surface.

4.3.3.1 Boundary Conditions at the Inlet

The condition for the velocity at the inlet is usually given via the setting of a velocity field. Numerical values can be implemented directly or a function programmed in the FORTRAN file USRBCS (CFX, 1997). Mathematically this is referred to as a Dirichlet boundary condition.

Turbulence quantities also need to be defined. The default formula for the turbulent kinetic energy in CFX is:

$$k_{in} = 1.5 \cdot (i_t \cdot u_{in})^2 \qquad (4.18)$$

Where $i_t$ is the turbulence intensity having a typical value of about 0.037 (in which case $k_{in} = 0.002 \cdot u_{in}^2$ in SI units), $k_{in}$ is the turbulence kinetic energy and $u_{in}$ the normal velocity at the inlet. This formula has been used for river flow in a recent paper by

Nguyen *et al.* (2000) from the University of Karlsruhe, although it remains an approximate formulation.

Other approaches have also been suggested:

(i) Celik, Rodi and Stamou (1986) used for a low entry velocity (velocity 22 cm/s):

$$k_{in} = 0.2 \cdot u_{in}^2 \tag{4.19}$$

Which is equivalent to (4.18), since the units are given in cm/s and $cm^2/s^2$ in (4.19).

(ii) Alfrink and van Rijn (1983) and Manson (1994) used a velocity log-law profile more appropriate for river systems:

$$k_{in} = \frac{u_*^2}{\sqrt{c_\mu}} \cdot \left(1 - \frac{z}{h}\right) \Rightarrow \overline{k_{in}} = 0.5 \cdot \frac{u_*^2}{\sqrt{c_\mu}} \tag{4.20}$$

Taking the Flood Channel Facility Series B (FCF) in example, (4.20) gives:

$$\overline{k_{in}} = 9.6 \times 10^{-4}$$

While (4.18) gives,

$$k_{in} = 3.1 \times 10^{-4}$$

The order of magnitude is nearly the same in this particular example. Moreover it is usually acknowledged that the accuracy of the boundary has little impact on the solution when using a rigid lid (Alfrink and Van Rijn, 1983; Ma Lin, 2000), which implies that either input would have a similar effect on the calculation further downstream from the inlet. However in the absence of detailed laboratory or field data (4.20) is usually preferred because it is more general and seemed better suited for open-channel flows.

CFX also requires a dissipation length scale. According to the manual a simple rule, but rather crude approach for large inlets, is to calculate

$$L = 4 \times Area/Perimeter \tag{4.21}$$

at the inlet. In the particular example of the FCF experiment reproduced hereafter this gives a length scale of 0.065. Ferziger and Peric (1996) however give a more general rule derived from dimensional analysis:

$$L = \frac{\nu_t}{c_\mu \sqrt{2k}} \tag{4.22}$$

Which in the particular example of the FCF experiment, leads to a length scale of 0.15, using the formula given in French (1985), based on Elder (1959) dispersion work for the calculation of $\nu_t$:

$$\begin{aligned} \nu_t &= \kappa u_* z\left(1-\frac{z}{h}\right) \\ \overline{\nu_t} &= 0.067 \cdot u_* h \end{aligned} \tag{4.23}$$

And the turbulent kinetic energy calculated above ($\overline{\nu_t} = 0.07 \cdot u_* h$ in Elder's original work). This formulation is preferred because it is more general and consistent with the thesis problematic than equation (4.21), which is suited to duct flow boundaries.

As far as turbulence dissipation is concerned, the following is used:

$$\varepsilon_{in} = \frac{k_{in}^{3/2}}{0.3 \cdot L} \tag{4.24}$$

This relationship is common in most popular CFD codes. Alternatively one can use:

$$\varepsilon_{in} = \frac{|u_*|^3}{\kappa z}\left(1-\frac{z}{h}\right) \tag{4.25}$$

combined with (4.20) as a more general version of (4.24) (Manson, 1994).

4.3.3.2 Boundary Conditions at the Outlet

At the outlet a mass flow boundary condition is implemented to set the mass flow rate equal to the mass which enters the domain at the inlet. To do so, Neumann boundary conditions are imposed. At the exception of the velocity, all transported quantities (e.g. $k$ and $\varepsilon$ ) are given a zero normal gradient. This is a standard approach when solving the Navier-Stokes equations.

$$\frac{\partial k}{\partial n} = 0 \text{ and } \frac{\partial \varepsilon}{\partial n} = 0 \tag{4.26}$$

In order to ensure mass-conservation (essential for a finite volume approach), the procedure for the velocity field is slightly different. In fact at all time, the mass flow into the domain must equal the mass flow out for an incompressible flow. Consequently, the normal velocity field gradient is first set to zero, the mass-conservation discrepancy calculated and corrected by setting the normal velocity gradient equal to a constant. The basic assumption behind this "mass-conservative" boundary condition is that of a fully developed flow.

$$\frac{\partial U}{\partial n} = \text{constant} \tag{4.27}$$

Although the above assumption (4.27) is one of the most commonly used approaches in treating outflow boundaries, it does possess some constraints. In theory, it is applicable only where fully developed flow exists, and might therefore be inadequate where vortices exist. The fully developed flow condition and the impact of the inlet/outlet boundary conditions on the solution need to be assessed to ensure that it does not affect the numerical solution.

4.3.3.3 Boundary Conditions at the Walls – Law of The Wall

At the walls a no-slip condition applies which means that the velocities tangential and normal to the walls are zero. Alternatively one could set a given value of the shear stress at the wall instead of a condition on the velocity, however this is usually not used for a solid wall boundary condition.

Close to the walls, the Navier-Stokes equations as presented in Chapter 2 would require a very fine grid to properly resolve the linear sub-layer and the turbulent boundary layer. This requirement is removed by replacing the grid by a model of the boundary layer: The wall function, Fig. 4.12. In the vicinity of the wall it is assumed that the fluid shear stress is equal to the wall shear stress, $\nu_t(\partial u_\tau/\partial n) = u_*|u_*|$, for which the local shear velocity $u_*$ is required.

In the laminar sub-layer of the boundary layer, the velocity condition is given as:

$$\frac{u_\tau}{u_*} = y^+ \tag{4.28}$$

Where $u_\tau$ is the tangential velocity, parallel to the wall, $u_*$ is the shear velocity, and $y^+ = yu_*/\nu$ is the non-dimensional distance normal to the wall. The transition between laminar and turbulent flow regions has been determined experimentally when $y^+ = 11.63$ (Chang, 1988).

In the turbulent shear flow region a general form for the law of the wall can be given as:

$$\frac{u_\tau}{u_*} = \frac{1}{\kappa}\ln(E(k_s^+)\cdot y^+) \tag{4.29}$$

Where, $E(k_s^+)$ is a function of the non-dimensional roughness height, $k_s^+ = k_s u_*/\nu$, in which $k_s$ is the roughness height, $\kappa$ is the Karman's constant taken equal to 0.4 here, and $\nu$ is the kinematic viscosity. The physical nature of the function $E(k_s^+)$ depends on the boundary condition itself. To determine the boundary conditions for the different components of the function $E(k_s^+)$, a criteria has been proposed by Schlichtling (1968):

(i) Hydraulically smooth:

$$0 \le \frac{k_s u_*}{\nu} \le 5$$

(ii) Transition:

$$5 \le \frac{k_s u_*}{\nu} \le 70$$

(iii) Hydraulically rough:

$$70 \le \frac{k_s u_*}{\nu}$$

Inside each sub domain, Chang (1988) gives an explicit account of the function $E(k_s^+)$, for at least the two extreme regions:

(i) Hydraulically smooth:

$$\frac{u_\tau}{u_*} = \frac{1}{\kappa}\ln(y^+) + 5.5 \Leftrightarrow E(k_s^+) = E = 9.0$$

(ii) Transition:

$$\frac{u_\tau}{u_*} = \frac{1}{\kappa}\ln(\frac{y^+}{k_s^+}) + f(k_s^+) = \frac{1}{\kappa}\ln(y^+) + \frac{1}{\kappa}\ln\left(g(k_s^+)\right) \Leftrightarrow E(k_s^+) = g(k_s^+)$$

(iii) Hydraulically rough:

$$\frac{u_\tau}{u_*} = \frac{1}{\kappa}\ln(\frac{y^+}{k_s^+}) + 8.5 \Leftrightarrow E(k_s^+) = \frac{30}{k_s^+}$$

Where $y^+ = yu_*/\nu$ is the non-dimensional distance from the wall. However, a function needs to be formulated for $E(k_s^+)$ in the transition condition.

In CFX, the hard-coded formula for the law of the wall is only valid for smooth surfaces (i.e. modelled as a constant equal to 9.0). A function that will extend the validity of the law of the wall beyond smooth surface boundary conditions and/or low turbulence models is consequently required, to amend the default formula via a user-Fortran subroutine in the file `USRWTM` (CFX 1997). This is not difficult for a hydraulically rough surface as the function is clearly defined by the theory (see (iii) above), and it has been successfully implemented in Launder and Spalding (1974) for example. However, a general function for $E(k_s^+)$, which covers the whole range of boundary conditions while remaining sufficiently robust to be added to CFX, needs to be formulated so that it can cover the entire range of $k_s^+$ values independently and provide more flexibility.

The difficulty lies into formulating a function $g(k_s^+)$ that is continuous and tends asymptotically towards 9.0 for small $k_s^+$ values and towards $30/k_s^+$ when $k_s^+$ becomes larger than 70. Mathematically we could consequently be looking at a function of the type $g(k_s^+) = \dfrac{a}{b + c\cdot k_s^+}$, with $\dfrac{a}{b} = 9.0$ and $\dfrac{a}{c} = 30.0$. The simplest function of this kind is:

$$g(k_s^+) = \frac{9.0}{1 + 0.3\cdot k_s^+} = \frac{E}{1 + 0.3\cdot k_s^+} \qquad (4.30)$$

As a matter of fact because of the function properties, one can generalise the above proposal to:

$$\forall k_s^+,\ E(k_s^+) = \frac{E}{1+0.3\cdot k_s^+} \tag{4.31}$$

The above function is very close to a proposal made by Naot (1984):

$$E(k_s^+) = \frac{E}{\left\{\dfrac{1+0.3k_s^+}{\left(1+\dfrac{20}{k_s^+}\right)}\right\}} \tag{4.32}$$

A similar approach has also been adopted in the commercial code FLUENT under the form $E(k_s^+) = \dfrac{E}{1+c\cdot k_s^+}$ ($c$ = constant), which strengthens the validity of the author's proposal for CFX.

Having established a function that spans continuously over the entire range of the boundary conditions and that tends asymptotically towards $E$ = 9.0 for smooth wall conditions and towards $30/k_s^+$ for rough wall conditions, one needs to evaluate its validity against physical data evidence. To do so it is decided to consider the experimental work of Nikuradse (1933) on roughness, and use the result of two studies that have attempted to fit a function to his data points.

A first approach to fit Nikuradse's data is presented by Cebeci and Bradshaw (1977):

$\forall k_s^+ \in [2.25;90.0],$

$E(k_s^+) = \exp[\kappa \cdot (B - \Delta B)] = \exp\left[\kappa \cdot \left(B - \left[B - 8.5 + \frac{1}{\kappa}\ln(k_s^+)\right]\sin\{0.4258 \cdot (\ln(k_s^+) - 0.811)\}\right)\right]$

$\forall k_s^+, k_s^+ > 90.0$

$E(k_s^+) = \exp\left[\kappa \cdot \left(8.5 - \frac{1}{\kappa}\ln(k_s)\right)\right]$

(4.33)

With $B = 5.45$. This has been used by Sinha *et al.* (1998) for example.

Sajjadi and Aldridge (1993) give a second similar approach:

$$\forall k_s^+,\ E(k_s^+) = \frac{\left(\exp\left[\kappa \cdot \left(\left(5.5 + \frac{1}{\kappa}\ln(k_s^+)\right)\exp[-0.062 \cdot \ln(k_s^+)] + 8.5 \cdot \{1 - \exp[-0.062 \cdot \ln(k_s^+)]\}\right)\right]\right)}{k_s^+} \qquad (4.34)$$

Both empirical functions are displayed on Fig. 4.13, together with the theoretically derived formulations proposed by the author and by Naot (1984). What can be seen on the figure is that: (i) such amendment as proposed by the author is necessary as $E$ varies considerably with $k_s^+$ and, (ii) although much simpler than (4.33) and (4.34) the thesis' proposal behaves very well in the low roughness region and from the middle of the transition region onwards. At the beginning of the transition region however, where $10 \le k_s^+ \le 40$, it under-predicts the value of the function $E$ derived from equations (4.33) and (4.34). In that respect Naot's (1984) proposal appears to be better, although it slightly over-predicts the data values in this region as well. The author's attention being on smooth surface flume (FCF) and, on rough river flows (Rivers Severn and Ribble) where $k_s^+$ tends to be larger than 50, proposal (4.31) was found to be similar enough to the empirical fitted curves to be judged adequate for the present investigation. However, in future applications the author will consider using Naot's formula. The latter appears to be more consistent with the data over the whole spectrum of $k_s^+$ values, and it can be derived in a more rigorous fashion from mathematical considerations, as shown by the author in his analysis leading to equation (4.30).

Consequently, in the CFX models presented hereafter the following law of the wall is applied:

$$\begin{aligned} &\frac{u_\tau}{u_*} = y^+ \quad \text{if } y^+ < y_0 = 11.63 \\ &\frac{u_\tau}{u_*} = \frac{1}{\kappa}\ln(E(k_s^+)y^+) \text{ otherwise} \end{aligned} \tag{4.35}$$

With $E(k_s^+)$ given in the table below. This equation is solved iteratively for $u_*$ using the velocity computed at the first internal grid point.

| Law of the Wall $\frac{u_\tau}{u_*} = \frac{1}{k}\ln(E(k_s^+)\cdot y^+)$ in CFX | |
|---|---|
| Hydraulically Smooth Wall Condition | $E(k_s^+) = E = 9.0$ |
| Transition and Hydraulically Rough Wall Conditions | $E(k_s^+) = \frac{E}{(1+0.3k_s^+)}$ |

**Table 4.1** – Law of the Wall modified in CFX

When the law of the wall is applied attention should also be paid to the position of the first node on the grid, close to the wall. A near wall flow is taken to be laminar if $y^+ < 11.63$. It is therefore important to ensure that the finite element grid does not encroach into this region, otherwise a transport equation validated for turbulent region will be applied in a region of laminar flow and a significant error will occur. This was systematically checked using the facility available in CFX 4.2 that provides the $y^+$ values for every node in the domain. However maintaining $y^+$ above 11.63 everywhere in the model is often impossible. In flows with recirculation at the wall, the velocity component parallel to the wall at the re-attachment point is zero, which means that the simulation reverts to the laminar case (Versteeg and Malalasekera, 1995). On the other hand one should also try to ensure that the first node is inside the log layer where the law of the wall is valid to obtain an accurate transition between the boundary layer and the turbulent flow velocity profiles, Fig. 4.12. Using the example of the FCF with grids FCF-1, FCF-2 and FCF-3 (Table 5.1), the first node was positioned at a distance $y^+ = 322$, $y^+ = 240$ and $y^+ = 168$ from the bottom wall respectively, based on an average shear velocity. This is not always achievable when modelling large-scale geometries, using relatively coarser meshes, and should be taken into account during the discussion of the results.

Turbulence quantities also need to be calculated at the walls. The turbulent kinetic energy is calculated in the control volume immediately adjacent to the wall. The boundary conditions on velocity at the walls are used to calculate $k$, and the dissipation term as follows, assuming local equilibrium:

$$k = \frac{u_*^2}{\sqrt{c_\mu}} \text{ and } \varepsilon = \frac{c_\mu^{3/4} k^{3/2}}{\kappa z} = \frac{|u_*^3|}{\kappa z} \tag{4.36}$$

4.3.3.4 Boundary Condition at the Wall – Determination of the Roughness Height

The previous section has established the theoretical and physical background for the law of the wall. It has shown that the formulation used in the present work relies on one variable ($y^+$) and one parameter ($k_s$, in $k_s^+$, which can take different values in different regions of the model). The latter determines the relative roughness of the wall surface(s) to which it is associated, and needs to be set. This is usually done during the validation/calibration phase of the model construction.

Several studies have attempted to investigate the physical meaning of this parameter. Yalin (1977) attributes three main causes to wall roughness (see Fig. 4.14):

(i) grain roughness;

(ii) bedform roughness;

(iii) roughness effects due to the presence of suspended material in the flow.

The latter will not be considered in the following investigation, as this study is not concerned with sediment transport. In addition to these three parameters the author would add turbulence effects at the walls. Quantification of these different effects remains difficult as evidenced by the widespread range of formulations available in the literature, see Whiting and Dietrich (1990) for example. As a result, with the current available knowledge roughness height mostly remains a calibration parameter, used to tune the model output to measured data.

Historically, relationships between roughness and particle diameters were investigated in smooth experimental channels to quantify $k_s$ on the ground that the most obvious form of roughness was that created by grain irregularities at the wall. Clearly this approach is limited in its application to real rivers. Nevertheless, this type of relationships has remained and has been implemented to relate roughness height to particle size in more complex channels. Classical formulations have been given as $k_s = 3.5 \times D_{84}$ or $k_s = 6.8 \times D_{50}$ (Clifford *et al.*, 1992), where $D_{XX}$ stands for the grain diameter for which *XX%* of the particles are finer; but there exists many variations of these formulas. Unfortunately, which momentum loss mechanism is included in such formulas in not really well known. The investigation of Clifford *et al.* (1992) has attempted to distinguish (i) from (ii) using large particles ($D_{50}$ = 40 mm), which has resulted in the calculation of different coefficients, e.g. quite logically for grain roughness, $k_g = 0.3 - 0.5 \times D_{50}$ (skin friction). The use of Clifford's results shows that the grain-roughness relationship is inadequate to determine the overall roughness height, as grain-related roughness seems to be causing low momentum losses. Formulas such as $k_s = 6.8 \times D_{50}$ therefore include several momentum loss mechanisms that are not well understood, nor quantified, resulting in larger roughness height values.

There is also a lot of uncertainty regarding the relevance of the above formulas outside the range of conditions for which they have been derived. One of the author's concerns is that most of the work that has led to the above formulations has been done for limited ranges of particle diameters. For example Whiting and Dietrich (1990) reported a range of values between 0.68 and 7.7 mm and seem to include all roughness effects shown on the previous page, while Clifford *et al.* (1993) used a mean diameter in the region of 40 mm. In particular formulas such as $k_s = 6.8 \times D_{50}$ or $k_s = 3.5 \times D_{84}$ have been derived for relatively small grains. Yet they are now applied by some CFD users without being questioned, leading to the use of large $k_s$ values such as 0.250 m (Hodskinson and Ferguson, 1998). Clearly the relevance of choosing roughness values should be discussed with more care, especially if the latter are also affected by the numerical solution. Numerical factors, such as inadequate discretization scale and turbulence closure assumptions in CFD models might in fact require high lumped $k_s$ values for the model to dissipate the correct amount of energy, in natural channels especially. Wall roughness could therefore be artificially increased in the model, and would only match the results of grain diameter roughness calculations "coincidentally", and not on physical grounds. More knowledge should be gathered to formulate $k_s$ in a more objective fashion, taking into account the different physical and numerical causes involved in its determination. This would be particularly valuable in natural channels, where skin friction is clearly not the only source of roughness.

It is therefore quite difficult for a modeller to attribute a roughness height to a given channel, because this is far from an "exact" science. Hydraulicians often prefer to use Manning's n roughness when it comes to open channel friction losses, although such a parameter is even more subjective to determine and dependent on the channel geometry. Although of less relevance to CFD, the next section is important because Manning's n is generally used by engineers when assessing how rough a channel is for one- or two-dimensional models.

Formulas exist to relate particle diameters to Manning's n values, and the latter to equivalent roughness height. One is derived from the HR Wallingford tables (Ackers, 1991) and yields:

$$k_s(mm) = (n/0.038)^6 \tag{4.37}$$

This only applies for a limited range of $k_s$ where the condition $10 \leq R/k_s \leq 100$ ($R$ = hydraulic radius) is satisfied, which confirms application to rough engineered canals and natural channels (Ackers, 1991).

A more general approach is therefore needed. Massey (1989) gives a theoretically derived equivalence:

$$k_s(SI) = 14.86 \cdot R / \exp_{10}\left(\frac{0.0564 \cdot R^{1/6}}{n}\right). \tag{4.38}$$

(4.38) is quite close to the derivation in Chow (1959):

$$k_s(SI) = 12.20 \cdot R / \exp_{10}\left(\frac{0.0457 \cdot R^{1/6}}{n}\right) \tag{4.39}$$

From (4.39) Strickler arrived at an average:

$$k_s(ft) = (n / 0.0342)^6 \tag{4.40}$$

Assuming a median grain-size. As reported in Chow (1959), (4.39) was successfully applied in the United States, and in particular on the Mississippi. Krishnappan and Lau (1986) also used (4.40) for their three-dimensional model of flood plain flow.

For the FCF experiment (4.37) yields a ridiculously small value of the order of a fraction of a micrometer. When $R/k_s$ is calculated for this experiment, it appears that it is 3 to 4 orders of magnitude larger than the upper limit! Consequently equations (4.38) to (4.40) are used and yield values in the range $2.0 \times 10^{-4}$ to $8.0 \times 10^{-4}$ m. This seems to fit reasonably well in the table given in French (1985) for simple, smooth materials or surfaces. However, and as underlined in Chow (1959), the above formulas were usually used to evaluate Manning's n values from the knowledge of $k_s$, for which the result n is quite insensitive. The converse is not true and leads to large ranges of roughness height values for small variations of Manning's n, especially when n is large.

4.3.3.5 Boundary Condition at the Free Surface

The free surface is in fact modelled as a rigid lid. This is a common approximation that is used to restrain the third dimension in order to define a fixed domain to be meshed. Considering only the most recent work (Demuren, 1993; Manson, 1994, Cokljat and Younis, 1995; Sinha *et al.* 1998) it is clear that it is the most popular approach. It has been found to work very well in the field of mechanical engineering where some correction was required on the dissipation term only (Naot and Rodi, 1982; Gibson and Rodi 1989).

Moreover, if such correction might be important for straight channel turbulence modelling, it is not certain that it would be significant in a natural meandering channel. Alfrink and van Rijn (1983) noted that the rigid lid assumption appears to be valid for small Froude numbers. In the current work, the pressure term on the lid is monitored to ensure that the rigid lid is well positioned and recreate the natural conditions as closely as possible. More details about how this is done are given in Chapter 5 and Chapter 6 in the calibration results.

The correct positioning of the lid has implications beyond the simple relation of free surface to pressure. If the lid were wrongly placed this would certainly affect the distribution of mass in the solution domain, and in turn the velocity field through the mass conservation equation. This is another reason to monitor the pressure term in the domain. The calculation of a correct pressure field is also related to the adequate definition of the fluid properties, which have less visible impact on the velocity field when it is driven by an inlet boundary condition. Lavedrine (1996) for example did not do so in her attempt to model the Flood Channel Facility. As a result her model generated spurious pressures or the order of $10^3$ Pa in part of the model, which would correspond to hydrostatic pressure head equal to 50% of the prototype water height. As a result she failed to obtain a proper pressure distribution on the lid, which will have subsequently impaired her overall solution

Recent reports seem to indicate however that the use of a lid could affect the solution more significantly than originally thought, in particular in the way that it reduces the sensitivity of the solution to the boundary condition. Ma-Lin (2000) conducted identical experiments for a simple straight channel inbank flow with a rigid lid model and a free-surface mixed flow model, and compared them against laboratory data. He found that if the main solution was quasi-identical, the rigid lid model was far less sensitive to the boundary conditions and also seemed to yield very localised erroneous velocities along some of the boundaries (walls). On the other hand, the free-surface model was very sensitive to the boundary conditions and very expensive to run. The author also experienced this when trying to model a straight channel inbank flow using CFX free-surface Volume of Fluid (VOF) algorithm to simulate a mixed flow process.

An intermediate alternative to the coarse rigid lid assumption and the refined mixed flow would be the use of adaptive meshing or re-meshing algorithms. Work is currently being undertaken (Tchamen and Kawahita, 1998) to develop adaptive-mesh or re-meshing techniques in order to allow for greater freedom regarding the motion of the free surface. This area of modelling seems to be a subject on its own at the moment due to the strain such freedom can put on the stability, convergence, numerical accuracy as well as on the computer power that is required. Stability is a particular sensitive issue because the Navier-Stokes equations being strongly convective in channel flows means that the domain might have to be highly modified (i.e. moved) in some regions between consecutive time steps. Deciding about the extent of the new domain is a difficult task, and one key question is which velocity to choose to "move" the mesh boundaries. If an adaptive-mesh solution is adopted, the mesh in the sensitive areas is stretched, which can be detrimental to the solution accuracy. If a re-meshing technique is chosen, heavy calculation work has to be conducted to interpolate the variables between the old and the new mesh. No satisfactory technique has been produced up to now, and using such methods in the current research programme would probably constitute an additional element of uncertainty.

Consequently, the free surface is modelled as an impermeable (zero normal velocity) wall, or lid, of constant shear (e.g. 0 if no wind). The turbulent terms are calculated as for the bed wall, from the shear values.

### 4.3.4 Solution Algorithms

#### 4.3.4.1 General Principles

The principles that are presented hereafter are quite generally applied in CFD, and can be found in codes such as CFX, FLUENT or FLOW-3D among others.

This section is needed because it underlines the principles of the solution method, which is usually conducted in two stages.

The Navier-Stokes equations are particularly complex to solve. Firstly the momentum equations contain non-linear convection terms and are intricately coupled. A second issue is that of pressure, which appears in the form of a gradient in all momentum equations, but does not possess a separate transport equation that would help determine its value. These are the keys to understanding the following principles:

(i) If one assumes that an initial velocity and pressure fields are known it is then possible to relax the non-linearity issue by calculating the velocity flux $\rho\vec{U}$ ($\nabla \cdot \left(\rho\vec{U} \otimes \vec{U}\right) = \rho\vec{U}\left(\nabla \cdot \vec{U}\right)$) in the momentum equations from that initial velocity field, and calculate the pressure in a similar fashion.

(ii) A new velocity field is then calculated from the terms left in the gradients and divergence operators (see equations (2.2) and (2.3)). In the case of an incompressible fluid if the initial pressure field that is applied is satisfactory, the new velocity field should then satisfy the continuity equation.

Both non-linearity and velocity-pressure linkage issues can be resolved if a proper iterative algorithm that calculates the initial fluxes and pressure fields adequately is implemented.

Algorithms such as SIMPLE (Patankar and Spalding, 1972) for example are designed for this purpose. The fluxes are calculated from "guessed" velocities, and so is the initial pressure, which is then corrected from a variant of the continuity equation for the pressure correction. The updated pressure is then used to correct the velocities. As a consequence of this process instabilities can occur due to the fact that the update between two consecutive solutions is too large for the inner iteration to be satisfied easily (see below). Consequently under-relaxation factors (URF) are applied to limit the growth of the correction. This calculation process is part of what has been described in point (i) above, and is called the outer iteration. More details are given below (4.3.4.2).

Point (ii) necessitates the implementation of an inner iteration process during which the now linearized momentum equations are solved (see 4.3.5). For efficiency reasons this is also an iterative process conducted at each outer iteration, until the reduction factor in the Navier-Stokes equation is below a certain percentage (given by the users) of the initial residual. To limit the number of inner iterations, it is recommended to allow only a maximum number of them to take place. However in difficult turbulent problems, a higher number of iterations is recommended on the turbulent terms and on the mass term compared to the velocities. The smaller the URF in the outer iteration the easier it is to conduct the inner iteration because the velocities in the new fluxes are closer to the previously calculated velocities that satisfied the algorithm. But it is also more time consuming as the changes are implemented more slowly. On the other hand the larger the URF, and the larger the change in velocities, the more difficult it might be to meet the reduction factor criteria. This is then translated into wiggles and possibly instability in the overall outer solution. This shows that a good balance between the two processes in necessary.

This two-stage iterative process is conducted until convergence or a satisfactory reduction in the residuals is achieved, see Section 4.3.6.

4.3.4.2 Pressure-Linkage Equations

Regarding pressure-linkage equations, the different standard SIMPLE techniques as well as PISO are available in CFX, SIMPLEC being the default algorithm. This equation is vital to ensure a proper pressure field in the model and justify the use of the rigid lid assumption (see 4.2.3). SIMPLE is known to be outperformed in terms of accuracy and efficiency by its more recent variants (SIMPLER and SIMPLEC). These have robust convergence characteristics in strongly coupled problems, but it is difficult to ascertain which of the two is superior to the other. Regarding PISO, Versteeg and Malalasekera (1995) report that in flows where the scalar variables are not strongly coupled to the velocities, this technique is faster than the SIMPLE variants. However this is not true in the opposite situation. In the present work SIMPLEC is used by default.

### 4.3.5 Numerical Solvers

There exist four families of matrix solver in CFX: (1) line relaxation; (2) conjugated gradient (CG); (3) Stone's Strongly Implicit Procedure; and (4) Algebraic Multi-Grid (AMG). During a simulation, each equation for each phase can be solved using a different technique if required.

#### 4.3.5.1 Line relaxation

This is a technique that is particularly suited to one-dimensional flow problems as it consists in an elimination procedure from one end of the equation system to the other end and is based on Thomas 1949 Tri-Diagonal Matrix Algorithm (TDMA, in Versteeg and Malalasekera, 1995). In two and three-dimensional flow problems this can only be done in a line-by-line fashion along the grid, and therefore the spread of boundary information into the domain is slow. This technique is also very sensitive to the flow direction, which is not known a priori in some domains.

Higher order schemes also involve nodal information other than that of immediate neighbours, which impairs efficiency and can be undesirable in terms of numerical stability. Where QUICK is used for example the TDMA needs to be upgraded to the Penta-Diagonal Matrix Algorithm (PDMA). In the case of boundary fitted and multiblock grids the need to incorporate a large number of contributions from neighbouring nodes limits the application of such a solution scheme.

#### 4.3.5.2 Stone's Strongly Implicit Procedure (SIP)

Stone's method (1968) was developed to offer an improved alternative to the classical iterative procedures such as the Jacobi techniques, or the alternating direction iteration method (ADI). ADI represents a significant improvement in terms of cost and convergence speed compared to the classical methods, but for problems with complex geometries or when its set of parameters is not properly defined it can still fail to converge. Stone's algorithm was initially produced to improve the ADI scheme limitations. It alters the problem matrix $A$ by the addition of a complement matrix $A'$, so that $A + A'$ can be factored in a simple lower-upper matrix product (LU), which can be easily manipulated. The iteration procedure is then written:

$$(A + A')X_{k+1} = (A + A')X_k - (AX_k - B) \tag{4.41}$$

If $\delta_{k+1}$ is the difference between $X_{k+1}$ and $X_k$ and $r_k$ is the residual from the $k^{th}$ iterate, the above equation can also be written:

$$(A + A')\delta_{k+1} = L(U \cdot \delta_{k+1}) = L(V_{k+1}) = r_k \tag{4.42}$$

Since the residual is known and $L$ is simple to invert, $V_{k+1}$ is easily obtained, which in turn leads to a solution for $\delta_{k+1}$ and the calculation of $X_{k+1}$. The simplicity of each calculation step ensures that the method requires less computation and storage space than classical methods, and is therefore very efficient. This method is particularly suitable for structured meshes.

4.3.5.3 Algebraic Multi-Grid (AMG)

The multi-grid method is directly founded on numerical analysis considerations and is related to the rate of convergence of the error. Typically, the algebraic equations are solved on the control volume grid so that the error is rapidly reduced locally. However, the global or smooth error is not or little reduced, which holds back convergence. Using Fourier analysis, it can be shown that the terms in the global error correspond to error components of wavelength $\lambda$ larger than the grid spacing $h$ that cannot be reduced on the local grid (see Quarteroni and Valli, 1994; Wright, 1987). As the mesh size decreases and the mesh complexity is increased, the gap between the components of the local error and the larger components of the smooth error is widening, and the rate of convergence deteriorates.

This difficulty could be overcome if the problem was defined in another space, on a coarser grid. In fact, the idea to increase the size of the discretization mesh artificially and reduce the local error more slowly while lowering the impact of the smooth error on the convergence is the key to multigrid. Properly speaking, multigrid is a "preconditioning" method, i.e. a technique that prepares the numerical problem for an optimum resolution using a given solution algorithm. The solution algorithm is in general similar to the Jacobi, Gauss-Seidel or CG methods described in 4.3.4 (this is the case in CFX). This preconditioning also presents the appealing characteristic that the relaxation sweep to reduce the smooth error is carried out on a coarser mesh, hence is faster. Originally, multigrid was literally implemented as a grid coarsening process relying on the structure of the original fine mesh (Brandt and Dinar, 1977). This "geometric" multigrid technique required an algorithm able to generate its own hierarchy of coarser grids and was quite restricted. Consequently Lonsdale (1993) proposed a simpler, more efficient and more robust method based on the use of the transformation matrix only, called algebraic multigrid (AMG). This algorithm used in CFX is summarised hereafter.

In order to solve $AX = B$ (where $A$ is a sparse $N \times N$ matrix) from the knowledge of an initial guess $X_l$ for $X$ on the fine grid, a solution $X^c_l$ for $X^c$ is sought in a coarser $M \times M$ space:

$$(KAK^T)\cdot X^c = B^c \tag{4.43}$$

With, $K$ the space transformation matrix, $X^c = K^T X$, $B^c = K\cdot(B - AX_l)$, and $M < N$. After one implementation, this leads to an improved solution:

$$X_{ll} = X_l + K^T X^c_l \tag{4.44}$$

The correction term $K^T X^c_l$ in (4.44) reduces the long wavelength error corresponding to the smooth error on the fine grid. The algorithm given by the above two equations is repeated in a recursive manner on a range of fine to coarse grids, until the grid can be coarsened no further.

The interested reader is referred to Trottenberg *et al.* (2001) for a complete and rigorous overview of multigrid, and in particular to appendix A for more AMG formulations. This recent reference states that multigrid methods are currently accepted as "*the fastest numerical methods for the solution of elliptic partial differential equations*" and "*among the fastest methods for many other problems, like other types of partial differential equations, integral equations etc.*". Because of the dynamic relation between preconditioning and numerical solution, such technique also holds the potential to be used as an adaptive grid scheme that would optimise an original grid for the resolution of a particular problem. More specifically, multigrid methods such as AMG are described as being extremely robust, and directly applicable to a vast range of CFD problems (geometric multigrid techniques generally require to be "tuned" for an application to different problems). They are also very effective to deal with complex geometries, which is appealing for an industrial use. However, AMG techniques require a higher availability of memory than their geometric equivalents, but this remains an accepted condition: "*most industrial users would prefer to wait longer for the results if they would otherwise not be able to solve what they really want to solve.*" (Trottenberg *et al.*, 2001).

4.3.5.4 Choice of Numerical Solvers

Stone's method and AMG were preferred because of their particular efficiency over the other methods. Stone's method is an enhanced version of the classic ADI method, and it was therefore felt that this would position the present research in the line of other current works and ensure the transition (see Table 3.1). The AMG solver has been reported to be particularly suitable and desirable for CFD application in natural open-channels (Sinha *et al.*, 1996; Mesehle and Sotiropoulos, 2000). Both teams made this recommendation after using ADI techniques. Sinha *et al.* (1996) expressed, twice, the need for multigrid methods in natural channel problems in their report final recommendations. In particular the implementation of such algorithm was suggested as a "*prerequisite*" when additional transport equations, e.g. for anisotropic turbulence or sediment transport, would be used (Sinha *et al.*, 1996, p. 162) and/or the channel geometry is complex. Using the AMG solver in comparison with Stone's method would therefore contribute to answering some of the questions recently formulated by fellow engineers.

The conjugate gradient method was not presented here, as it was not explicitly used with CFX during the course of this research work. It is presented in detail in section 4.4.4, as it is the central solver in TELEMAC.

### 4.3.6 Convergence Criterion

Owing to the size of the grids used in this study, as well as the cells' large aspect ratio, the convergence rate was expected to be slow. The usual criteria based on the reduction of the mass residual to $10^{-4}$ to $10^{-6}$ usually implemented in a finite volume problem could not reasonably be applied for reasons of time and storage requirements.

Two criterion were consequently chosen by the author:

(i) Reduction in magnitude of the residuals by at least three to four orders of magnitude (Sinha *et al.*, 1996, Meselhe and Sotiropoulos, 2000);

(ii) Minimal changes in the flow, turbulence and pressure field in the region of interest.

Typical histories of convergence are shown for two simulations run with the $k$-$\varepsilon$ and RSM turbulence models, using the AMG solver (Fig. 4.15 to 4.18). In Fig. 4.15, one can see the rapid convergence rate of the velocities due to the reduction of the local error occurring in less than 150 iterations for the $k$-$\varepsilon$ case. The smoothing of the global error is then more progressive and it takes about 1600 iterations to be satisfactorily reduced. In Fig. 4.16, it is clear that the pressure and turbulent quantities converge faster than the velocities. As observed by Sinha *et al.* (1996), the rate of convergence of the turbulence quantities is particularly good compared to the pressure and the velocity errors. This simulation corresponds to 45 hours of CPU time on a Sun Ultra 10/433 station with 384 MB of RAM.

Figs. 4.17 and 4.18 show a similar convergence history for the RSM. The convergence of the velocity term here is slower, especially for the U-momentum term. Convergence of the other terms such as mass and turbulence quantities is fast, and the latter in particular show an excellent convergence. This simulation required about 60 hours of CPU time on the Sun Ultra 10 station, and despite the unsatisfactory U-momentum convergence rate, the simulation was stopped since little variation in the velocity field at the bend was observed. On Figs 4.17 and 4.18 the peak that is observed at 100 iterations is due to the initialisation of the Reynolds stress terms. In fact, the simulation is started with the $k$-$\varepsilon$ model, from which calculated quantities are then used to initiate the RSM variables after 100 iterations.

### 4.3.7 Scalability

During the implementation of the Flood Channel Facility test case with CFX (Chapter 5) records were kept about the CPU performance of the AMG solver for different levels of grids and different turbulence models. These results are plotted on Fig. 4.19 after 2000 iterations, and show that as the mesh density increases the increase in CPU is approximately parabolic, which implies that greater detail in the numerical solution will be very expensive. However no record were obtained for larger grids than CFX FCF-3 (see Chapter 5) on our machine since the problem was becoming too large to handle beyond 250,000 elements. At this stage the square terms in both trendline formulas is not much larger than the linear terms, which explains why below 250,000 elements the curvature of the trendline is mild.

Another comment of importance is that even if it seems that the outcome of a simulation using the RSM for turbulence is about 20-25% more expensive to obtain, the reader should bear in mind that Fig. 4.19 is plotted for all simulations after 2000 outer iterations. With the $k$-$\varepsilon$ model this was sufficient to reduce the residuals by four orders of magnitude (Figs. 4.15 and 4.16), however this is hardly the case with the RSM (Figs. 4.17 and 4.18). The RSM is consequently <u>at least</u> 25 to 30 % more expensive to run, and this is for relatively simple problems. In fact as the RSM is more prone to stability problems, in particular as the complexity of the problem increases, it is often necessary to reduce parameters such as the under-relaxation factors (URF) and reduction factors, and increase the number of inner iterations on the turbulence terms to ease the convergence process. This further increases the computation time.

In the above examples, CFX performed very well because the geometry is fairly simple. For comparison, the performance of the AMG solver is plotted on the same graph for the river applications containing about 180,000 elements (Chapter 6) after the residuals have been reduced by 3 to 4 orders of magnitude. The gradient of the lines plotted for these on Fig. 4.19 is between 2.3 and 2.6, i.e. 10 times larger that the linear component of the lines plotted for the FCF. This illustrates how expensive it is to apply a general three-dimensional flow solver to natural river geometry using a structured grid.

## 4.4 TELEMAC NUMERICAL ISSUES AND MATHEMATICAL ASSUMPTIONS

### 4.4.1 Construction of the Geometry and the Mesh

#### 4.4.1.1 The Geometry

The initial surface geometry is built automatically from the knowledge of the data points. It can be modified manually by the creation of structural lines that will enhance ground surface features such as bank lines. The two-dimensional base of the code is reflected in its surface nature: Only the bottom geometry is originally created for the entire domain. The third dimension is created at each time-step as the water height is calculated from the resolution of a two-dimensional Navier-Stokes problem on the bottom grid, and used to elevate layers of prisms from knowledge of the surface mesh (Fig. 4.20). In TELEMAC there is no clear distinction between geometry and mesh, and little is said about the determination of the geometry by the developers. In fact it seems that both processes are inter-connected, e.g. as the structure lines used to shape the geometry also act as constraint lines for the mesh. These lines should be used to stress the main topographical features of the geometry in order to structure the grid. It is particularly important that they are used when the grid is coarse in order to ensure that sharp changes in the topography (riverbank and bottom lines for example) are well captured by the numerical grid.

#### 4.4.1.2 Mesh Construction

TELEMAC uses a two-dimensional mesh, or to be more exact layers of two-dimensional meshes. It is constructed using a pre-processing software called MATISSE, to yield an unstructured triangular grid based on a Delaunay triangulation. The unstructured nature of the mesh is very useful to represent complex topographical profiles. The database for the first triangulation is a series of $(x,y,z)$ coordinates that can be directly downloaded from survey instruments.

As a result, most of the discussion is on the adequate design of the bottom layer, which can be seen as the vertical projection of the future three-dimensional mesh. The controls to design the mesh are basic:

(i) the use of constraint lines to shape the geometry and the mesh;

(ii) the choice of local element-size criterion.

Constraint lines are used to give a little more of structure to the mesh and enhance the definition of the bottom layer. They are to be positioned in areas where a sharp change in the profile gradient occurs, to avoid spurious topographical interpolation and be able to construct a body-fitted mesh, or where local refinements is needed such as on the banks. This type of refinement on sharp variable gradients is essential to ensure that there is no spurious free-surface interpolation (see Fig. 4.21 for an illustration) and that adequate volumes of water are calculated in the domain. The choice of local element size values enable to refine part of the mesh in areas of interest and progressively increase the mesh size further away. As a result, more economical grids can be designed, and problems run more efficiently.

As far as the vertical representation is concerned, it is adjusted at each time step after calculation of the water depth via a two-dimensional solver. The user simply chooses the number of layers (two surfaces border one layer) that he wishes to use, the first layer representing the bottom and the last the free surface. As a result a two-layer set up is a minimum. The default algorithm entails equally spaced layers over the depth of the calculated water column. However, a progressive spacing algorithm can be implemented in the Fortran subroutine. Fixed values can also be set. The fact that the third dimension is represented via layers is problematic in the sense that, in the case of a two-stage channel, very different aspect ratio cells are created on the flood plain and in the deeper channel. This consideration also means that the wish to have a fine series of elements close to the bed can generate elements that are too small on the flood plain, which can trigger numerical errors and mass-balance problems (a classical difficulty in finite elements). This probably means that an optimum number of layers might have to be chosen, maybe at he expense of the representation of the wall effects.

In general there is less control over the definition of the geometry and determination of precise element sizes in TELEMAC than in CFX. The scale of the problem to be tackled with TELEMAC would explain the more global nature of the control that is provided to the user to set the element size. The automatic generation of the bottom surface, as well as the mesh adaptive third dimension over the vertical, are valuable assets in fluvial hydraulics.

#### 4.4.1.3 Mesh Independence

This procedure is independent of the code, and is therefore conducted as described in 4.2.1.3. However the notion of mesh independence for TELEMAC is a controversial issue, since there appears to be a correlation between roughness and grid resolution due to the code's two-dimensional background (Hardy *et al.*, 1999; Hankin *et al.*, 2000), and because of the level of uncertainty in large scale models. It would probably be more adequate to talk about the "mesh impact" on the solution.

### 4.4.2 Numerical Discretization

TELEMAC relies on an operator-splitting technique to solve the Navier-Stokes equations:

(i) solution of a hyperbolic problem (convection) followed by;

(ii) solution of a parabolic problem (diffusion).

The hyperbolic problem is solved directly by the Method of Characteristics (MOC) or finite element methods (SUPG or MURD, Janin *et al.*, 1997a), while the parabolic problem is formulated as a finite element variational process. Both problems are solved consecutively at each time step.

#### 4.4.2.1 Discretization of the Convection Terms

A first approach is to treat the hyperbolic problem using the Method of Characteristics (MOC). This is the default in TELEMAC. It has good physical properties (monotonicity and upwind), interesting in advection-dominated flows, and is the fastest by far. However it can be quite diffusive and is not good at achieving mass-conservation, particularly on coarse meshes. This can pose problems for the treatment of the continuity equation for the advection of the water depth for which it is therefore not recommended.

An alternative for the advection of depth notably is to treat the first part of the equation using Petrov-Galerkin methods ( $W_l \neq N_l$ ) of the form $W_l = N_l + \alpha \cdot \Delta x/2 \cdot (\nabla \cdot N_l)$. If the constant $\alpha = 1$, the scheme can be shown to be equivalent to a first order upwind scheme. In TELEMAC an unconditionally stable discretization is available in the form of the Streamline Upwind Petrov-Galerkin formulation (SUPG, Brookes and Hughes, 1982). For stability reasons the weight function is de-centered in the direction of the flow by the Courant number, $Cr = |\vec{U}| \cdot \Delta t / \Delta x$, instead of $\alpha$, so that $W_l = N_l + \Delta t / 2 \cdot \vec{U} \cdot (\nabla \cdot N_l)$. The additional term gives more weight to the elements towards which the front is moving, and reduces greatly the occurrence of numerical diffusion compared to the other available methods in the code. It is interesting for the advection of depth to enhance mass conservation in the continuity equation. It is however a difficult scheme to use for the velocity and does not seem to be recommended by TELEMAC developers unless the diffusion with the MOC is too large.

Mass conservation for these two schemes can be improved by implementing sub-iterations, which will refine the quality of the convection velocities. This can be an interesting alternative when using the MOC in order to combine its particular time efficiency with iterative corrections to adjust the velocity field for each "outer" time iteration. It should also be noticed that although upwind has a poor reputation in finite difference this is not the case in finite element if it is used consistently. This is demonstrated in an example by Dick (1996) where the order of accuracy is unaffected by the use of upwind.

MURD schemes (Janin *et al.*, 1997a) are also available to the user in TELEMAC. These schemes are similar to a finite volume method that would be discretized in a finite element frame, i.e. equations (4.11) and (4.12) with unit weight functions but the variables discretized using (4.7) inside each element. By construction these schemes are mass conservative. They are therefore recommended for the tracers in TELEMAC.

4.4.2.2 Discretization of the Diffusion Terms

This stage is carried out using a finite element variational statement similar to that shown in 4.2.1.1, but only applied to the diffusion step equation (4.58). There is no alternative option for this part of the equation in TELEMAC. More details about the construction of the full variational statement can be found in Janin *et al.* (1997a, pp. 43-46). The general principle remains similar to that in equations (4.10) and (4.12) however.

4.4.2.3 Choice of Discretization Schemes

The choice of a finite element approach and the structure of the solution algorithm in TELEMAC dictate the schemes to use. In a way either one treats the problem using finite element for the whole transport equations, or one considers the properties of the Navier-Stokes equations to speed up the calculation process. In the latter case one can choose to obtain an approximate solution to the convection part of the Navier-Stokes equations using the MOC (for the velocities) prior to solving the parabolic part of the problem using a more accurate variational formulation. By default this approach is implicitly recommended for the velocities. On the other hand mass conservation is better handled using SUPG for the calculation of the advection of water depth in particular when the problem is diffusive. This imposes the nature of the scheme to be used for the depth.

Fortunately different approaches can be used simultaneously for the different variables during the same calculation. It is therefore recommended to solve the convection terms for the velocity and turbulence fields using the MOC, use SUPG for the advection of the flow depth (Horritt, 2000, Hankin *et al.*, 2000) and treat the passive tracers with the MURD (Janin *et al.*, 1997a). This is what will be done in the forthcoming work, unless stated otherwise.

### 4.4.3 Boundary Conditions

In general the choice and implementation of the boundary conditions in TELEMAC is simpler than for a general fluid dynamics code since they are designed to tackle open-channel hydraulics cases. This of course also implies that the application of the TELEMAC system is limited to such cases, but this is already assumed from the start in the way the Navier-Stokes equations are simplified and the solution calculated.

#### 4.4.3.1 Open Boundary Conditions

In fluvial hydraulics numerical modelling it is recommended to use a velocity field at the inlet and a fixed water height at the outlet (Bates *et al.* 1998). Consequently, Dirichlet boundary conditions are implemented at the inlet and outlet nodes for the main variables $\vec{U}$ and $h$.

However, since it is customary to have information in the form of a discharge $Q$ at the inlet or a rating curve ($h = f(Q_{outlet})$) at the outlet for river systems boundary conditions "pre-conditioners" can be implemented in the model to convert such information into a velocity field-water height format. In particular the rating curve type of boundary can be implemented because of the adaptive nature of the mesh on the vertical in the TELEMAC system to deal with unsteady hydraulic conditions

The discharge inlet condition Q3D (Janin *et al.*, 1997a, Example 1) is simple to implement. Based on knowledge of the initial or previous time step water height, the wetted area at the inlet is integrated using a trapezoidal integration between two boundary bottom nodes. A mean velocity is calculated and applied in a uniform manner over the width and height. It is obvious here that poorly defined boundaries (i.e. with few nodes) will yield inaccurate velocity fields. In addition, because of the integration of the finite element problem, inlet boundaries including large dry areas can also be a source of instability. This is because even if part of the boundary is in reality dry, it is not really viewed as such from a finite element perspective. In fact each node possesses a water height, possibly small or even slightly negative, to enable an integration over the entire domain. If Q3D is confronted with large dry boundaries it will attempt to set a velocity field even in very shallow areas which can then result is spurious large velocities, which will damage the solution. It is therefore recommended to write an additional line of programme to scan the boundary and implement Q3D in wet regions only.

At the outlet, the rating curve boundary condition was implemented in a similar fashion by the author. The subroutine is called Q3DSORTIE. It scans the outlet boundary nodes and detects those that are "wet", ranking them from $k$ to $k+w$. As shown on Fig. 4.21 it then integrates each inter-node wetted area in turn, multiplied by the mean depth-averaged velocities:

$$\forall i \in [k; k+w-1]_{\text{Outlet BC}}$$

$$A_{i+\frac{1}{2}} = 0.5 \cdot d_{i+\frac{1}{2}} \cdot (h_i + h_{i+1}) \tag{4.45}$$

$$\vec{V}_{i+\frac{1}{2}} = 0.5 \cdot (\vec{V}_i + \vec{V}_{i+1}) \tag{4.46}$$

$$Q_{i+\frac{1}{2}} = A_{i+\frac{1}{2}} \cdot \vec{V}_{i+\frac{1}{2}} \cdot \vec{n} \tag{4.47}$$

Where $\vec{V}_i$ and $h_i$ stand for the velocity field and water height at node $i$, $A_{i+\frac{1}{2}}$, $\vec{V}_{i+\frac{1}{2}}$ and $Q_{i+\frac{1}{2}}$ respectively stand for the wetted area, the mean velocity field and the mean calculated discharge between nodes $i$ and $i+1$. All of these are calculated at time $n$. The downstream discharge $Q^n_{outlet}$ is obtained as the sum of the $Q^n_{i+\frac{1}{2}}$ and reformulated for stability reasons as:

$$Q^n_{outlet} = \theta \cdot Q^n_{outlet} + (1-\theta) \cdot Q^{n-1}_{outlet} \tag{4.48}$$

With $\theta > 0.5$.

This is then implemented in the field rating curve to yield the water surface level $S$:

$$S^{n+1} = f(Q^n_{outlet}) \tag{4.49}$$

From the knowledge of the bottom topography $Z_f$, each boundary node water height at time $n+1$ is then determined:

$$h_i^{n+1} = S^{n+1} - Z_{f,i} \tag{4.50}$$

It usually works reasonably well, but might require some adjustment of the parameter $\theta$ to remain stable.

As far as the turbulence quantities are concerned (for the $k$-$\varepsilon$ model only), the inlet conditions are set internally, making use of the local equilibrium assumption, as:

$$k = \frac{u_*^2}{\sqrt{c_\mu}} \qquad \varepsilon = \frac{u_*^3}{\kappa \cdot k_s} \tag{4.51}$$

These conditions are calculated from the domain shear velocity and stem from (4.23) and (4.25), which have also been shown to be similar to the conditions implemented in CFX. The turbulent eddy viscosity is calculated using:

$$\nu_t = c_\mu \frac{k^2}{\varepsilon} \tag{4.52}$$

At the outlet, a von Neuman boundary condition is set as:

$$\frac{\partial k}{\partial n} = 0 \text{ and } \frac{\partial \varepsilon}{\partial n} = 0 \tag{4.53}$$

Which is identical to (4.26) in CFX.

4.4.3.2 Boundary Conditions at the Walls

A no-slip boundary condition is set at the walls, to model the friction and impermeability effects at and through the wall. To model the development of the velocity boundary layer in the region immediately above the wall, a law of the wall is implemented. The latter depends on the turbulence model that is chosen, mixing-length or $k$-$\varepsilon$.

In the case of a mixing-length model, a simple law of the wall is implemented as:

$$C = \frac{1}{\kappa} \cdot \ln\left( \frac{\Delta Z}{k_s / 4} \right) + 4.9 \tag{4.54}$$

With $\Delta Z$ corresponding to the distance between the first node and the bottom and $C$ is a constant (Janin *et al.*, 1997a). In this case the fluid shear stress is taken equal to a quadratic function of velocity of the form $\nu_t(\partial u_\tau / \partial n) = u_\tau |u_\tau| / C^2$, which is a two-dimensional approach common in quasi-3D hydraulics codes (Manson, 1994). The bed shear stress is implicitly chosen as a parabolic function of the horizontal "layered" velocity, which makes it inadequate for the calculation of detailed bed shear stress strongly influenced by up- and down-welling effects, and more sensitive to roughness (Lane *et al.*, 1999).

In order to relate (4.54) to classical formulations such as those shown before, a simple comparison is carried out for the Flood Channel Facility (FCF) flume modelled in the next chapter. If $k_s$ = 0.2 mm, (4.54) yields a ratio of 13.9 at 2 mm from the walls. If $k_s$ = 0.8 mm a ratio of 10.5 is calculated at the same location. If $k_s$ = 0.2 mm (4.31) gives 12.6, and if $k_s$ = 0.8 mm 10.1 at 2 mm from the bottom. The relation given by (4.54) is for rough boundary hydraulics, but is relatively smoother than the relationship implemented in CFX (4.31). However, in the particular case of the FCF experiment, considering a shear velocity of the order of 0.0244 m/s, the above difference would entail a maximum difference of 2.5 mm/s (about 0.65% of the mean velocity) at 2 mm from the wall.

TELEMAC-3D being a French code predominantly used in estuarine and coastal hydraulics the default formulation uses Chézy $C$ for roughness. Roughness height and Chézy C values are related using Ramette formula in TELEMAC-3D (Janin *et al.*, 1997a):

$$C = \frac{105.6}{k_s}\left(\frac{k_s}{4 \cdot h}\right)^{1/24} \cdot h^{1/6} \qquad (4.55)$$

The $k$-$\varepsilon$ model uses a more classic form for the law of the wall (see section 4.3.3.3). It simply applies the Launder and Spalding (1974) equation and assumes that the fluid shear stress is equivalent to the bed shear stress, as in CFX. At the wall the equilibrium between production and dissipation of turbulent kinetic energy is made, and the boundary conditions set as:

$$k = \frac{u_*^2}{\sqrt{c_\mu}} \qquad \varepsilon = \frac{u_*^3}{\kappa \cdot k_s} \qquad (4.56)$$

There is less flexibility to implement and access to information related to the law of the wall in TELEMAC compared with CFX. This is generally true for all of the boundaries as pointed out in the manual (Janin *et al.*, 1997b). The code has clearly been designed for river and estuarine hydraulics, as the hard-coded law of the wall formulation illustrates.

4.4.3.3 Boundary Conditions at the Free Surface

A standard impermeable condition is applied to reduce the velocity normal to the wall to zero. In the absence of wind a zero-shear boundary conditions is applied at the free surface. If the velocity and intensity of the horizontal wind is known, these are applied as boundary conditions.

### 4.4.4 Solution Algorithm

The solution algorithm in TELEMAC usually relies on an operator-splitting technique (Benqué, Haughel and Viollet, 1982; Quarteroni and Valli, 1994)). This classical approach enables the solution of the equations in two stages, using an intermediate solution.

4.4.4.1 Convection

TELEMAC first calculates an initial guess from a partial equation made of the unsteady and convection terms. This is called the convection step:

$$\frac{\vec{U}_C - \vec{U}^n}{\Delta t} + \vec{U} \cdot \nabla(\vec{U}) = 0 \tag{4.57}$$

Where, $\vec{U}_C$ represents the so-called "convection velocity", solution of (4.57). $\vec{U}^n$ represents the velocity at the previous time step, denoted $n \cdot \Delta t$. Equation (4.57) is explicit in time. However, the problem is that $\vec{U}_C$ and $\vec{U}^n$ are not calculated on the same mesh. In fact the mesh is upgraded at the end of each time step in TELEMAC, when the new water height is re-calculated. Taking this change into consideration is important to account properly for the convection effect on the free surface. The problem must therefore be set on an independent frame of reference *(x*, y*, z*)*, which fixes the mesh in time and space. In fact this new reference consists in writing the vertical referential component in a time-independent manner, i.e. a form that is free of the free-surface variation impact, while the other time and space dimensions remain unchanged. More details about this transformation can be found in Janin *et al.* (1997a).

4.4.4.2 Diffusion

Using the partial solution obtained from the convection stage, the propagation-diffusion part of the equation is taken as:

$$\frac{\vec{U}_D - \vec{U}_C}{\Delta t} - \nabla \cdot \left(\nu \cdot \nabla\left(\vec{U}\right)\right) = 0 \qquad (4.58)$$

It is then formulated in the fixed mesh as a finite element variational problem by an integration process similar to that shown in equation (4.12). This calculation is quite heavy and the interested reader is referred to Janin *et al.* (1997a) for further details about matrix calculations. Several sub-iterations can be requested by the user to enhance the calculation of the non-linear problem.

4.4.4.3 Propagation-Conservation

In TELEMAC-3D the propagation step is carried out as follows (Hervouet and van Haren, 1996):

$$\frac{\vec{U}^{n+1} - \vec{U}_D}{\Delta t} - \frac{1}{\rho}\nabla(p) - \vec{Source} = 0 \qquad (4.59)$$

The depth-averaged version of the above equation is solved first using TELEMAC-2D to yield water depth values at each node, thus enabling the computation of an updated three-dimensional mesh. Once this is this is done equation (4.59) is used to calculate the three-dimensional values of the horizontal components of the velocity field (because hydrostatic pressure is assumed, the momentum equation for the vertical flow component is indeed reduced to the relationship between pressure and water depth). This calculation is direct as it consists of a simple solution of a linear combination of vectors (Janin *et al.*, 1997a, p. 50).

Finally the vertical component of the flow is calculated by closure on the mass-conservation equation. In fact to facilitate the computation this is carried out on the fixed frame of reference, since the true vertical component of the flow is only required at the next convection step. A consequence of this procedure is that "*trespassing the hydrostatic assumption in 3D results in degraded solutions with spurious vertical velocities*" (Hervouet and van Haren, 1996)

More details on the whole numerical procedure are given in Janin *et al.* (1997a).

### 4.4.5 Numerical Solvers

The reason for using the operator splitting technique lies in the possibility it offers to choose the most efficient solver for each of the stages. The solution of (4.56) followed by a simplified finite element problem is generally more efficient than that of a complete variational matrix. It also enables the use of linear shape functions for both velocity and depth, assuming there is little numerical diffusion.

Equation (4.58) is a hyperbolic equation that is easily solved by the method of the characteristics, the solution for such an equation corresponding to finding $U_i^{n+1}$ along a characteristics curve from the knowledge of $U_i^n$. In order for $U_i$ to remain constant on the characteristics curve:

$$\begin{aligned} \frac{dx^*}{dt} &= U_x = U \\ \frac{dy^*}{dt} &= U_y = V \\ \frac{dz^*}{dt} &= U_z^* = W^* \end{aligned} \tag{4.60}$$

This determination of the characteristics is made using a Runge-Kutta method with an explicit velocity field fixed at time $n$, i.e. $U = U^n$, $V = V^n$ and $W^* = W^{*n}$ in the fixed frame of reference mentioned in 4.4.4.1.

Using this two-step approach simplifies the iteration of the problem matrix $A$ by reducing it to a symmetric diffusion matrix $A'$ after the first step (elimination of the "convection-matrix" component, see Hervouet and Van Haaren, 1995). This matrix is also definite positive. It is particularly suited to Conjugate Gradient (CG) methods, and in particular the Generalized Minimal Residual (GMRES) method. In reality however the sparse matrix $A'$ is not fully assembled but treated in parts using the Element-By-Element technique (EBE, Hervouet, 1991) for further iteration economy.

The CG resolution algorithms are all presented in numerical analysis handbooks, an example of which is Quarteroni and Valli (1994). These relaxation methods are inspired by the Jacobi method described hereafter. In order to solve the problem $A'X = B$, $A'$ is split into $A' = P - N$, where $P$ is a suited non-singular preconditioning matrix. From the knowledge of an iterate $X_k$, $X_{k+1}$ is defined as $PX_{k+1} = NX_k + B$. If the residual from the $k^{th}$ iteration is then defined as $r_k = B - A'X_k$, and the $\delta_{k+1}$ is the difference between $X_{k+1}$ and $X_k$, then $\delta_{k+1} = P^{-1}r_k$ and $X_{k+1} = X_k + P^{-1}r_k$. This is more generally written:

$$X_{k+1} = X_k + \alpha_k \cdot r_k \tag{4.61}$$

Where $\alpha_k$ is an acceleration parameter. When the residual is small enough, the algorithm is stopped. This method is one of the most elementary iterative methods available and is not expected to perform well except for simple problems for which its cost is not excessive.

The classic CG method was initially designed to produce an exact solution of a matrix problem in which $A'$ is assumed to be symmetric, definite positive. It is in principle slightly different to the Jacobi method. In the CG method (Hestenes and Stiefel, 1952), the update of the iterate is obtained by replacing $r_k$ by another "direction" $p_k$ such that:

$$\forall k \in \mathbb{N},\ \langle p_{k+1}, A'p_k \rangle = 0 \tag{4.62}$$

With $p_0 = r_0$ (the $p_k$ are said to be "$A'$-conjugate".).

Then,

$$X_{k+1} = X_k + \alpha_k \cdot p_k \tag{4.63}$$

The $\alpha_k$ is a scalar calculated from the knowledge of $r_k$ and $p_k$. This method is based on the fact that solving the problem $A'X = B$ using an iteration process such as (4.63), when $A'$ is symmetric positive definite, is equivalent to minimizing $\phi(w) \equiv (A'w, w) - (B, w)$ using an iterative polynomial solution of the form:

$$w = X_0 + \sum_{j=0}^{k} \beta_j \cdot p_j \quad \text{with} \quad \beta_j \in \mathbb{R} \tag{4.64}$$

Which is achieved if, and only if the $p_j$ are A'-conjugate (Quarteroni and Valli, 1994).

This iteration technique is used in TELEMAC, to obtain very accurate results in a small number of steps. If the problem is solved differently (e.g. finite element applied to the complete equation) and $A$ is not symmetric-positive, a variant called the Bi-Conjugate Gradient (Bi-CG) method is also available in the code. Similarly, if the solution proves difficult to reach the availability of GMRES offers a robust alternative to the standard CG to treat the problem. The type of preconditioning $P$ that has been found to be most efficient in TELEMAC is the Crout preconditioning (Janin et al, 1997a).

### 4.4.6 Convergence

There is no steady state option as such in TELEMAC, since it has been specifically designed to model unsteady flow situations. However, it is possible to set stop criterion concerning the magnitude of the variation of the main variables such as the velocities. Additionally, integrating the discharge on the downstream boundary using Q3DSORTIE can help verify that the flow has reached steady state. Once such an observation has been made the user must also ensure that the overall mass conservation inside the domain is reasonably good.

### 4.4.7 Scalability

Because TELEMAC is a hydraulic code destined to large-scale fluvial problems scalability was investigated using various model constructed for a practical case, that of the Ribble reach in Chapter 6. It was indeed judged more appropriate to illustrate the code's scalability in its field of application. For references the times necessary to the calculation carried out in Chapter 5 for the Flood Channel Facility (FCF) using extremely fine grids will be given.

Both the outputs of the two- and three-dimensional data for the simulation of an observed 100-$m^3$/s flow over a 20 min. period are considered, Figs 4.22 and 4.23. Two interesting features are immediately noticeable:

(i) CPU time seems to increase parabolically in two dimensions, although the time increase is dominantly linear in most of the best fit line equation shown on Fig. 4.22.

(ii) On the other hand CPU time increases in a quasi-linear fashion in three dimensions (based on data collected with a maximum number of nodes below 20,000), Fig 4.23.

Point (ii) indicates that the closure over the vertical dimension is quite efficient, especially as the number of nodes in the domain increases. However, this is probably related to an enhanced calculation of the mass term improved by a finer resolution, therefore requiring fewer iterations, which compensate for the cost of the extra nodes. In some cases a balance could even be met between a low-resolution model and a higher resolution model that eventually proves more accurate. A similar justification is also likely to be valid in two dimensions where the trend is quasi linear. These types of features are very good, as the cost of a numerical solution is not going to increase significantly beyond the proportional cost of adding these extra nodes.

Using the data obtained from the FCF experiments using TELEMAC-3D, a similar result was found: the grid with 62,300 elements took 14.5 hours to run whereas the finer grid (130,290 elements) took about 26.5 hours. It should be noted that in this case, it is more expensive to run the FCF model with TELEMAC than it is to run it in steady state with CFX. It is however completely different in practical river flow situations where TELEMAC is more adapted and will yield an answer – albeit in two-dimensional layers – more efficiently, e.g. in under 25 hours for a model of the Severn with 72,675 elements.

## 4.5 REFERENCES FOR CHAPTER 4

1. Ackers, P., (1991), "Design Manual for Straight Compound Channels", HR Wallingford Report, Wallingford, UK.
2. Alfrink, B.J., van Rijn, L.C., (1983), "Two-equation Turbulence Model for Flow in Trenches", J. Hydr. Eng., Vol. 109, No. 3, pp. 941-988.

3. Anderson, J.D., (1995), "Computational Fluid Dynamics – The Basics with Applications", McGraw-Hill, New York, USA.
4. Benqué, J.-P., Haughel, A., Viollet, P.-L, (1982), "Numerical Models in Environmental Fluid Mechanics", in "Engineering Applications of Computational Hydraulics – Homage to Alexandre Preissmann", Vol. 2, M.B. Abbott and J.A. Cunge (eds.), Pitman.
5. Bradbrook, K.F., Lane, S.N., Richards, K.S., (2000), "Numerical Simulation of Three-Dimensional, Time-Averaged Flow at River Channel Confluences", Water Res. Res., Vol. 36, No. 9, pp. 2731-2746.
6. Brandt, A., Dinar, N., (1977), "Multigrid Solution to Elliptic Flow Problems", in "Numerical Methods in Partial Differential Equations", S. Parter (ed.), pp. 53-147.
7. Cebeci, T., Bradshaw, P., (1977), "Momentum Transfer in Boundary Layers", Hemisphere Publishing Corporation.
8. CFX, (1997), "CFX 4-2: Solver Manual", AEA Technology plc, UK.
9. Celik, I., Rodi, W., Stamou, A.I., (1987), "Prediction of Hydrodynamic Characteristics of Rectangular Settling Tanks", in "Turbulence Measurements and Flow Modeling", C.J. Chen, L.-D. Chen and F. M. Holly Jr (eds.), Proc. Int. Symp., Iowa, USA.
10. CFX Users' Manual (1997), AEA Technology plc, Harwell, Oxfordshire, UK.
11. Chang, H.H., (1988), "Fluvial Processes in River Hydraulics", Krieger.
12. Chow, V.T., (1959), "Open-Channel Hydraulics", McGraw-Hill.
13. Clifford, N.J., Robert, A., Richards, K.S., (1992), "Estimation of Flow Resistance in Gravel-Bedded Rivers: A Physical Explanation of the Multiplier of Roughness Length", Earth Surface Processes and Landforms, Vol. 17, pp. 111-126.
14. Cokljat, D., Younis, B.A., (1995), "Compound Channel Flows: A Parametric Study using a Reynolds-Stress Transport Closure", J. Hydr. Res., Vol. 33, No. 3, pp. 307-320.
15. Demuren, A.O., (1993), "A Numerical Model for Flow in Meandering Channels with Natural bed Topography", Water Resources Res., Vol. 29, No. 4, pp. 1269-1277.

16. Dick, E., (1996), "Introduction to Finite Element Techniques in Computational Fluid Dynamics", in "Computational flui Dynamics – An Introduction", J.F. Wendt (Ed.), Springer, pp.230-268.
17. Dick, E., (1983), "Accurate Petrov-Galerkin Methods for Transient Convective Diffusion Problems", Int. J. Num. Meth. Eng., Vol. 19, pp. 1425-1433.
18. Elder, J.W., (1959), "The Dispersion of Marked Fluid in Turbulent Shear Flow", J. Fluid Mech., Vol. 5, pp. 544-560.
19. Ferziger, J.H., Peric, M., (1996), "Computational Methods for Fluid Dynamics", Springer.
20. Finnie, J.I., (1994), "Finite-Element Methods for Free-Surface Flows", in "Computer Modeling of Free-Surface and Pressurized Flows", M. Hanif Chaudhry and L.W. Mays (Eds.), NATO-ASI Series, Kluwer Academic Publisher.
21. French, R.H., (1985), "Open-Channel Hydraulics", McGraw-Hill International Editions, Civil Engineering Series.
22. Gaskell, P.H., Lau, A.K.C., "Curvature Compensated Convective Transport: SMART, a New Boundedness Preserving Transport Algorithm", Int. J. Num. Meth. In Fluids, Vol. 8, pp. 617-641.
23. Gibson, M.M., Rodi,W., (1989), "Simulation of Free Surface Effects on Turbulence with a Reynolds Stress Model", J. Hydr. Res., Vol. 27, No. 2, pp. 233-244.
24. Hankin, B.G., Hardy, R., Kettle,H., Beven, K.J., (2001), "Using CFD in a GLUE Framework to Model the Flow and Dispersion , Earth Surface Process. and Landforms (to appear).
25. Hardy, R.J., Bates, P.D., Anderson, M.G., (1999), "The Importance of Spatial resolution in Hydraulic Models for Flood Plain Environments", J. Hydr., Vol. 216, pp.124-136.
26. Harlow, F.H., Welch, J.E., (1965), "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", The phys. of Fluids, Vol. 8, No. 12, pp. 2182-2189.
27. Hervouet, J.-M., van Haren, L., (1995), "TELEMAC-2D Version 3.0 – Principle Note", Report HE-43/94/052/B, EDF-DER, LNH Chatou, France.

28. Hervouet, J.-M., van Haren, L., (1996), "Recent advances in Numerical Methods for Fluid Flows" in Flood plain Processes, M.G. Anderson, D.E. Walling and P.D. Bates (1996), John Wiley & Sons Ltd.
29. Hestenes, M.R., Stiefel, E.L., (1952), "Method of Conjugate Gradients for Solving Linear Systems", J. Res. NBS, Vol. 49, pp. 409-436.
30. Hodskinson, A., Ferguson, R.I., (1998), "Numerical Modelling of Separated Flow in River Bends: Model Testing and Experimental Investigation of Geometric Controls on the Extent of Flow Separation at the Concave Bank", Hydr. Processes, Vol. 12, pp. 1323-1338.
31. Horritt, M.S., (2000), "Calibration of a Two-Dimensional Finite Element Flood Flow Model Using Satellite Radar Imagery", Water Res. Res., Vol. 36, No. 11, pp. 3279-3291.
32. Idelsohn, S.R., Oñate, E., (1994), "Finite Volumes and Finite Elements: Two 'Good Friends'", Int. J. num. Meth. Eng., Vol. 37, pp. 3323-3341.
33. Janin, J.M., Marcos, F., Denot, T., (1997a), "Code TELEMAC-3D Version 2.2 – Note Théorique", Report HE-42/97/049/B, EDF-DER, LNH Chatou, France (In French).
34. Janin, J.M., David, E., Denot, T., (1997b), "TELEMAC-3D Version 2.2 – User's manual", Report HE-42/97/051/B, EDF-DER, LNH Chatou, France.
35. Krishnappan, B.G., Lau, Y.L., (1986), "Turbulence Modeling of Flood plain Flows", J. Hydr. Eng., Vol. 112, No. 4, pp.251-266.
36. Launder, B.E., Spalding, D.B., (1974), "The Numerical Computation of Turbulent Flows", Comput. Methods Appl. Mech. Emg., Vol. 3, pp. 269-289.
37. Lavedrine, I., (1996), "Evaluation of 3D Models for River Flood Applications", Report TR 6, HR Wallingford, Wallingford, UK.
38. Lee, J.K., Froelich, D.C., (1986), "Review of Literature on the Finite Element Solution of the Equations of Two-Dimensional Surface Water Flow in the Horizontal Plane", U.S. Geological Survey Circular 1009, U.S. Geological Survey, Denver, CO.
39. Lonsdale, R.D., (1993), "An Algebraic Multigrid Solver for the Navier-Stokes Equations on Unstructured Meshes", Int. j. Num. Meth. Heat Fluid Flow, Vol. 3, pp.3-14.
40. Ma-Lin, (2000), *personal communication*.

41. Manson, J. R., (1994), "The Development of Predictive Procedure for Localised Three Dimensional River Flows" PhD Thesis, The University of Glasgow.
42. Massey, B.S., (1989), "Mechanics of Fluids", Sixth Edition, Chapman and Hall.
43. Mesehle, E.A., Sotiropoulos, F., (2000) "Three-dimensional Numerical Model for open-Channels with Free-Surface Variations", J. Hydr. Res., Vol. 38, No. 2, pp. 115-121.
44. Naot, D., (1984), "Response of Channel Flow to Roughness Heterogeneity", J. Hydr. Eng., Vol. 110, No. 11, pp. 1558-1587.
45. Naot, D., Rodi, W., (1982), "Calculation of Secondary Currents in Channel Flow", J. Hydr. Div., Vol. 108, No. HY8, pp. 948-968.
46. Nguyen, V.T., Nestmann, F., Eisenhauer, N., (2000), " Three Dimensional Computation of River Flow", Proc. Fourth Int. Conf. On Hydroinformatics, Iowa City, July 2000, CD-Rom.
47. Nikuradse, J., (1933), "Stromungsgesetze in Rauhen Rohren", Forschrg. Geb. D. Ing.-Wesens, Heft, p. 361, (In German).
48. Patankar, S.V., Spalding, D.B., (1972), "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flow", Int. J. Heat and mass Transfer, Vol. 15, pp. 1787-1806.
49. Quarteroni, A., Valli, A., (1994), "Numerical Approximation of Partial Differential Equations", Springer Verlag.
50. Rhie, C.M., Chow, W.L., (1983), "Numerical Study of the Turbulent Flow past an Airfoil with Trailing Edge Separation", AIAA, Vol. 21, pp. 1527-1532.
51. Roache, P.J., (1989), "Need for Control of Numerical Accuracy", J. Spacecraft and Rockets, Vol. 27, No. 2, pp. 98-102.
52. Sajjadi, S.G., Aldridge, J.N., (1993), "Second moment Closure Modelling of Turbulent Flow over Sand Ripples", Proc. Fifth Int. Symp. On Refined Flow Modelling of Turbulent Flows, Paris.
53. Schlichtling, H., (1968), "Boundary-Layer Theory", McGraw-Hill.
54. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.

55. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1996), "Numerical Model Studies for Fish Diversion at Wanapum/Priest Rapids Development – Part I: Three-Dimensional Numerical model for Turbulent Flows through Wanapum Dam Tailrace Reach", IIHR Limited Distribution Report No. 250, Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa, USA.
56. Stone, H.L., (1968), "Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations", SIAM Journal on Numerical Analysis, Vol. 5, Issue 3, pp. 530-558.
57. Tchamen, G.W., Kahawita, R.A., (1998), "Modelling Wetting and Drying Effects over Complex Topography", Hydr. Proces., Vol. 12, pp. 1151-1182.
58. Thompson, J.F., Warsi, Z.U., Mastin, C.W, (1985), "Numerical Grid Generation", Elsevier Science Publishing.
59. Trottenberg, U., Oosterlee, C., Schüller, A., (2001), "Multigrid", Academic Press.
60. Versteeg, H.K., Malalasekera, W., (1995), "An Introduction to Computational Fluid Dynamics – The Finite Volume Method", Longman.
61. Whiling, P.J., Dietrich, W.E., (1990), "Boundary Shear Stress and Roughness over Mobile Alluvial Beds", J. Hydr. Eng., Vol. 116, No. 12, pp. 1495-1511.
62. Wright, N.G., (2001), *personal communication.*
63. Wright, N.G., (1987), "Multigrid Solutions of Elliptic Fluid Flow Problems", PhD Thesis, The University of Leeds, UK.
64. Yalin, M.S., (1977), "Sediment Transport", Pergamon Press.

# Chapter 5:

# Numerical Models Evaluation

## – The Flood Channel Facility Test Case

# Chapter 5:

# Numerical Models Evaluation

## – The Flood Channel Facility Test Case

"*If one does not know to which port one is sailing, no wind is favourable.*"

(Seneca; 4 BC- AD 65)

"*Each problem that I solved became a rule, which served afterwards to solve other problems.*"

(René Descartes; 1596-1650)

## 5.1 RATIONALE

As presented in the introduction, the aim of this research is to investigate the application of three-dimensional modelling and Computational Fluid Dynamics (CFD) to natural two-stage channels with overbank flows. A quasi-three-dimensional finite element code, TELEMAC, and a CFD finite volume code, CFX, were selected; TELEMAC because of its applicability to fluvial problems and CFX because of possibilities offered by this CFD research tool in terms of numerical schemes, turbulence models and solvers. TELEMAC possesses a large validation set related to fluvial and estuarine applications, whereas CFX does not have a track record of application in this field. In general little work has been carried using CFD to simulate open-channel flow situations. Consequently, it was necessary to establish the feasibility of modelling overbank flows in open-channels using CFX and TELEMAC prior to attempting to reproduce the complex velocity fields measured in the field by Lancaster University. A detailed laboratory experiment was chosen from the Flood Channel Facility (FCF) experiments as a reference case for three-dimensional modelling.

Several reasons can be cited to justify the choice of the FCF experiment as an evaluation test case:

(i) The FCF *raison d'être* is the investigation of two-stage channels and flood hydrodynamics in a controlled environment. It matches exactly what constitutes the core of this research programme at a smaller scale.

(ii) The FCF Series B experiments with meandering two-stage channels resulted in fully three-dimensional velocity fields and are therefore well suited as test cases for CFX.

(iii) The choice of a simple, however meaningful, geometry enables the simplification of the two-stage flow problem, and truly assess the codes numerical techniques. This is a key issue since it is expected that modelling a natural channel would entail added complexity to the solution process, because of the natural river layout and irregularities.

(iv) The density and quality of the data in the FCF experiments (velocity and turbulent fields) exceeds by far that which can be measured in a natural channel, especially during a flood event.

The aim of this investigation is to assess the codes with respect to modelling flood flow hydrodynamics and set the degree of confidence that can be placed in them. Beyond the evaluation of the codes what is also sought is the assessment of novel CFD techniques applied to open-channel flows, e.g. the CCCT scheme or multigrid techniques as recommended by Sinha *et al.* (1996). This more specifically applies to the general CFD code CFX, whereas TELEMAC employs simpler techniques because of its specific industrial background (large-scale fluvial hydraulics) and offers fewer options. This is one of the reasons why this section will investigate the capability of CFX capability in more detail.

## 5.2 FLOOD CHANNEL FACILITY – EXPERIMENT B23

### 5.2.1 Experimental Set Up

The FCF is a 50-m long by 10-m wide flume, in which a 48-m long sinusoidal-shaped main channel is built. The main channel wavelength is 12 m, which means that 4 meanders are constructed. It is 150 mm deep and presents a trapezoidal cross-section with a top width equal to 1200 mm and a 45° side bank slope. The flood plain is 10 m wide and its slope is $9.96\times10^{-4}$. Details of the plan view for one meander are visible on Fig. 5.1 and a picture of the channel set up is shown in Fig. 5.2.

The FCF experiment simulated here is B23 (SERC FCF Series B Report, 1993). The total flow depth in experiment B23 is 200 mm, which means a 50-mm depth on the flood plain. This leads to a depth ratio coefficient equal to:

$$Dr = \frac{\textbf{Floodplain depth}}{\textbf{Total Depth}} = \frac{50}{200} = 0.25 \qquad (5.1)$$

The corresponding discharge is equal to 0.25 m$^3$/s.

The channel roughness is rather smooth compared to what one would expect in nature, and an overall skin friction value of 0.0105 is usually taken for Manning's $n$ (Lorena, 1993).

### 5.2.2 Description of the Flow

The FCF experiments are concerned with the detailed study of open-channel velocity fields during flood events. These events result in velocity fields that are highly three-dimensional, mostly due to the interaction of flood plain and inbank water flowing in different directions (Fig. 3.3). In turn, this has implications on the mechanics of sediment entrainment and deposition, and on pollutant mixing and dispersion.

What happens when overbank flows interact with inbank water flowing in the meandering main channel is the formation of strong helical currents due to the overbank flow passing over the main channel. The main channel flow is "rolled" inside the confines of the channel in a direction determined by the direction of the overbank flow. This phenomenon is different to what is usually observed for inbank flows, in which the faster surface water plunges as it hits the bank, e.g. at a meander bend. Wherever overbank and inbank flows are re-aligned the secondary current declines, but is still dominant. If the bend is sharp enough and the relative orientation of the two flows with respect to one another changes radically, overbank water shears against the inbank streamwise and secondary currents and starts to reverse the helical motion. Where this process occurs there is no dominant lateral current in the channel, and the overbank flow plunges inside the channel. On the contrary, where secondary current effects are stronger (e.g. at cross-section 8), the overbank flow tends to roll over the inbank helicoid, and accelerates it at the same time. This implies that there exists a very strong mixing and entrainment process inside the main channel. On the other hand, there is little mixing between inbank and overbank waters flowing in the central band of the FCF flume.

Another noticeable feature in the planview is the direction of travel of the flow. Where the *flood plain* flows passes over the main channel or where some inbank water is ejected onto the flood plain in the region of the cross-over, the flow travels at an angle with respect to the FCF flume centre line. This trajectory is skewed in the direction of the water flowing in the straight junction between two bends, and located immediately upstream. Consequently the water flowing on the flood plain does not travel following the steepest flood plain slope, but in a sinusoidal trajectory centred around the flume centre line, with variations occurring at each cross-over.

### 5.2.3 Location of the Data Collection

During the FCF experiments, all data were collected at and around the third meander (bend 6), at twelve cross-sections (Fig. 5.1). In order to demonstrate the helical motion past the bend, it was decided to compare results at four cross-sections, namely 1, 3, 5 and 8. Cross-section 3 is an obvious choice since it is located at the bend. Cross-section 5 was chosen because it lies in the region where the sense of fluid rotation is reversed for overbank flows. In addition, this is the area where the flood plain flow is plunging into the main channel, as illustrated in Fig. 5.3. Cross-section 8 was also used as it corresponds to the middle of the cross-over region, where the secondary velocities are expected to be at their strongest. Comparisons at these four sections should enable confirmation that the computer model has captured the reversal of the rotation past the bend, as well as, the evaluation of the secondary current intensity relative to the main channel flow.

Turbulence data were also collected during experiment B23. Reynolds stresses were measured in detail at cross-section 3, in planes normal to the cross-section, as well as turbulent velocity variations in all directions. These data are clearly presented in the SERC FCF manual (1993). Additional hand-written data were found at the end of the manual in the form of variations of Reynolds stresses over the depth in the vicinity of cross-section 8. These were collected along a section passing through the middle of cross-section 8, but parallel to the flood plain walls, called 8b (Fig. 5.1). They represent variations of the Reynolds stresses in the cross-sectional plane, as well as in the vertical plane normal to the section. The value of these two sets of data is mostly comparative in the scope of the current evaluation exercise. However, the fact that data are available at the cross-over and at the apex of the channel meander could be useful in order to understand and compare some of the hydrodynamic mechanisms.

### 5.2.4 Experimental Data

The FCF experimental data reported in the SERC FCF Series B Report (1993) were plotted with the following conventions. Velocity measurements were taken as a vector velocity norm (Fig. 5.4) and the angle made by the velocity vector with respect to the normal to the cross-sectional plane (Fig. 5.5). The deviation from the normal (0°) is then given using a trigonometric direction rule, positive to the left, negative to the right (Fig. 5.1, cross-section 5). The velocities are given in cm/s and the angles in degrees.

At Cross-section 1 there is a core of higher velocities in the main channel located between the cross-section centre and the left bank. It shows that the main flow is already positioned along the inner bank. Above bankfull higher velocities are visible on the flood plain and in the upper part of the main channel where the fluid contracts to move onto the left flood plain. The contribution of the upstream flood plain flow from the right is also clear, as it forms a closed isovel with the inbank core. Finally, in the main channel the isovels are parallel to the side walls, which shows that the inbank flow is mostly driven by the main channel geometry, but this pattern vanishes above bank level as the flood plain flow crosses the main channel. Considering the angle plot, the existence of a counter clockwise helicoid in the main channel is clearly seen. Close to the bend, the flow is deviated to the right (especially on the right bank side), and this trend is progressively reversed as the flow depth increases until it is fully deviated to the left. This rotating effect appears in the way the gradient of the angle is mostly parallel to the bed, and the angle lines are distorted in the upper right direction. Above bankfull, one can see that on the left bank the flow is quasi-perpendicular to the flood plain walls (angle close to 20-30 degrees, which is the cross-section angle). On the other hand the flow that arrives from the right is travelling at an angle of 25 degrees with respect to the flood plain side walls. This is cancelled by the main channel, which effectively re-align the flow.

At cross-section 3 the highest velocity water is flowing along the inner bank, and the isovel pattern is mostly parallel to the bank walls. This indicates that there is little interaction between inbank and overbank flows. This is confirmed on the angle plot: Above bankfull it seems that the flow is slightly deviated to the left in the main channel, but nearly parallel to the flood plain walls on the flood plain. The existence of the counter-clockwise rotating cell is seen on the angle plot which presents the same quasi-horizontal gradient of the angle (sloped in the upward direction to the right) along the depth in the main channel. The absence of flood plain contribution to increase the rotation effect in the main channel probably means that the recirculation is weaker. This would explain why the angle lines close to the bed are less distorted to the right than for cross-section 1.

Cross-section 5 shows the flow separation past the bend (40-cm/s isovel). The velocity core is crossing over towards the next inner bank. The distorted shape of this isovel on the right indicates the water plunging from the left flood plain into the centre of the main channel. The existence of a narrow region of high velocities along the left bank confirms this idea, because it must correspond to the plunging jet being redirected at the bed and rolling on the channel left side. On the angle plot the most remarkable feature is the vertical angle gradient, highlighted by a quasi-vertical angle line of –15 degrees in the channel centre. The existence of such a constant angle over the vertical indicates that there is little or no rotation of fluid in the centre of this cross-section. On the other hand the existence of a small positive angle at the bottom left corresponds to the birth of the counter-clockwise motion initiated by the flood plain plunging flow (see Fig 5.3). On the left flood plain (upstream), it can be concluded that the flow is nearly parallel to the flood plain walls, since cross-section 5 is located at an angle of 40 degrees with respect to the normal to the flood plain walls. The gradient of the angle across the bank region on the left bank is indicative of the strong interaction between the flood plain and inbank flows. On the other hand a sharper gradient on the right bank indicates that the water that is ejected from the main channel onto the flood plain is rapidly redirected in a direction with a 10-degree angle to the flood plain walls. The water past this cross over tends to travel in a direction that is slightly skewed to the left.

Cross-section 8 clearly shows the reversal of the flow rotation as well as its cause. On the velocity plot the strong impact of the flood plain flow is visible as a horizontal isovel exists across the channel above bankfull. The centre of the main channel at mid-depth is the slowest region, whereas very fast velocities occur along the walls and at the surface. This means that there exists a strong rotational structure inside the main channel, which is driven by the flood plain flow. In fact the angle plot reveals the existence of such a structure as the angle gradient is varying horizontally with depth, slightly falling towards the right. This points at a clockwise rotating structure. The fact that there is little angle variation on the left bank indicates that the water from the flood plain is shearing very strongly and rolling over the main channel rotation cell. The angle on the left flood plain also indicates that the upstream flow reaches the main channel in a direction that is nearly parallel to the flood plain walls. On the right bank on the other hand, the water that is ejected onto the flood plain changes direction very rapidly. The fact that the maximum deviation on the right is of the order of –50 degrees indicates that the water is also travelling in a slightly skewed direction to the left (+10 degrees). This confirms the deviation observed at cross-section 5.

The helical motion (Fig. 5.3) described by Ervine *et al.* (1994) and Willetts and Hardwick (1993) is clearly highlighted on the angle plots. At cross-section 1, it strongly influences the angle pattern by generating a negative angle region in the right corner, which is stretched in the upward direction along the wall. At cross-section 3, it occupies the central part of the main channel, which translates into a centred negative angle cell at the bottom and a horizontal angle gradient. At cross-section 5, the birth of the next helicoid through shearing effect results in a positive angle region along the left wall. This develops until it occupies most of the channel at the cross-over, at cross-section 8. The sign of these inbank angle isolines also indicates the dominant rotating direction, negative to the left, positive to the right.

### 5.2.5 Turbulence Experimental Data

Data related to the turbulent velocity variations at cross-section 3 are plotted on Figs. 5.6 to 5.8. Although these are of no use to evaluate the codes they are relevant to understanding the flow dynamics at the apex. From Figs. 5.6 and 5.7 it is clear that there is a strong correlation between the location of the highest turbulent fluctuations and that of the maximum isovels. Along the left bank velocity fluctuations in the normal and lateral directions can be as high as 15% of the local velocity. There is less fluctuation of vertical velocity however its core is also located along the left bank and it represents 9% of the main velocity (Fig. 5.8). Finally, the velocity difference between inbank and flood plain flows generates large velocity fluctuations in the vertical plane at the left bank/*flood plain* interface.

The second set of data available at section 3 is shown on Figs. 5.9 and 5.10. These plots of the local Reynolds stresses show the difficulty of taking and analysing such complex turbulence measurements. A neater post-processing interpretation of the data by the FCF experimentalists is reproduced on Fig. 5.11 and 5.12. Five main regions are visible concerning the distribution of the horizontal Reynolds stresses, $T_{yx}$ (Fig. 5.11). Two regions of high shear (-0.50 N/m$^2$) are located at the bed and along the left bank where most of the water is flowing in the streamwise direction. This simply corresponds to the shearing of the fast inbank flow against the slower overbank water and the walls. At the interface there is vertical distribution of the $T_{yx}$ in a V-shape reaching –0.25 N/m$^2$. Towards the centre of the cross-section, the Reynolds stresses are negative again and form a circular distribution around –0.15 N/m$^2$. The distribution of the vertical Reynolds stresses is more difficult to describe (Fig. 5.12). Its structure is more complex and the stress values are lower in magnitude and positive (0.20 N/m$^2$). In general large positive values of $T_{xz}$ are located along the walls, especially at the bottom and along the right hand side wall. These correspond to the interaction of the anticlockwise flow motion with the walls. A large radial structure appears in the top right corner of the main channel, which reaches values around 0.25 N/m$^2$. Along the left bank, Reynolds stresses are lower and indicate little vertical interaction.

At cross section 8b (Figs. 5.13 and 5.14), large values of the Reynolds stresses are visible (magnitude 0.60 to 0.70 N/m$^2$). This is expected since velocity data indicates a very strong horizontal and vertical interaction between the flood plain flow and the inbank water flowing at a 30-degree angle with it. In particular Fig. 5.13 indicates the horizontal interaction between the over-topping flow and the inbank flow. Fig. 5.14 highlights the vertical shear distribution along the bank/flood plain interface where the flow has not yet started to plunge.

### 5.2.6 Bed Shear Stresses

Plots of measured bed shear stress values (Fig. 5.15) were produced for experiment B23 cross-section 3 (SERC, 1993, Vol. 7(A)). A more complete set of data was collected for the so-called "semi-natural" channel, and is displayed in the same volume. They show a sharp increase of the bed shear stress on the outer bank from locations such as section 3 to the cross-over region. At the same time, the stress along the inner bank decreases slightly. The interested reader is referred to SERC Vol. 7(A) (1993).

### 5.2.7 Conclusions

The previous sections discussing results from FCF experiment B23 have presented:

(i) The interpretation of the detailed experimental velocity measurements through the formation of a helicoid at a channel bend due to overbank flows in a meandering compound channel;

(ii) The precise location of the sequence of events leading to the reversal of the helical velocity structure along the bend;

(iii) The relative size of this structure and the overall three-dimensional nature of the flow;

(iv) Detailed turbulence data at two important locations, where the main channel and flood plain flows travel in the same direction and, at the cross-over, where the overbank flows is expected to interact most strongly with the inbank flow.

The above analysis is consistent with the analysis and description provided by the FCF experimentalists (Lorena, 1992; Willetts and Hardwick, 1993; Ervine *et al.*, 1994) and other laboratory researchers (Shiono and Muto, 1998) for compound meandering channels with a depth ratio of 25%. In general, this confirms that the FCF should constitute a relevant benchmarking test case for assessing the applicability of three-dimensional Computational Fluid Dynamics (CFD) techniques for simulating flows in natural rivers.

## 5.3 MODEL OF THE FCF USING TELEMAC-3D

### 5.3.1 Preliminary Comments

This model has been constructed to obtain an initial image of simulating FCF data using a computer model, as well as providing a base for comparison against simulations obtained using the more sophisticated CFD techniques available in CFX. It is not expected to produce an accurate representation of the flow since the code has not been designed for detailed fluid mechanics predictions (i.e. it uses a hydrostatic pressure assumption and a layer approach over the vertical). Such a coding choice by the developers simply reflects the reality of TELEMAC's application field, in which fluid mechanics accuracy is not sought *per se*, but an industrial applicability for large-scale flow problems, in river and coastal environments, is.

The above reasons lead to considering single set-ups for the model using TELEMAC, and favour the quality of the discretization at the expense of the representation of turbulence, which is also believed to be secondary in the flow simulation presented hereafter (see 5.4.9). Since a preliminary investigation of the FCF was previously carried out at HR Wallingford (Lavedrine, 1996 and 1997), the following work also constitutes a continuation of this early research.

### 5.3.2 Spatial Discretization

5.3.2.1 Grid Construction

An unstructured surface mesh of 6,230 elements is built over the whole length of the FCF channel, Fig. 5.16 (TELEMAC FCF-1). A constant element size criteria is used to generate a regular mesh, unless constraint lines are used to restraint this condition, e.g. on the banks and in the main channel where constraint lines parallel to the bank lines are implemented. These constraint lines are equally spaced between the bank lines and the bottom lines. Five of these are used at the bottom and one on the banks. This is to reinforce the density of nodes in the regions where strong velocity gradients are expected to occur, and place nodes where the laboratory data were collected. The total number of nodes in the plan view is 3,243, and 10 layers of the plan view grid are replicated over the water column (62,300 elements and 32,430 nodes) to be as close as possible to the 0.15-m vertical interval used for the laboratory measurements. These are updated whenever the continuity equation adjusts the water height using the two-dimension solution module.

Each element of the above mesh covers an area of 0.084 m$^2$ ($h$ = 0.29 m) on average, but the typical element area is about 0.010 m$^2$ ($h$ = 0.10 m) on the banks and 0.040 m$^2$ ($h$ = 0.20 m) in the main channel, and larger than 0.160 m$^2$ on the flood plain ($h$ = 0.40 m, Fig. 5.16). Data were collected every 0.05 m on the banks and every 0.10 m in the channel. This resolution is that of Lavedrine (1996). She reports using no more than 35,000 elements for her model of the FCF using TELEMAC, although she only built a 5-layer model whereas data were collected in 13 locations over the vertical. As a result it is difficult to determine whether her model reproduce the correct features over the vertical or not, especially as the layer-averaged approach dominates the output (Figs. 60 to 62, Lavedrine, 1997).

For comparison a second mesh (TELEMAC FCF-2) was built, using 130,290 elements in three dimensions. This was achieved by increasing the resolution along the side walls and in the main channel, using smaller criteria and additional constraint lines. The new grid provides a resolution that is as fine as that of the measurements, i.e. an interval of $h$ = 0.10 m in the main channel and $h$ = 0.05 on the bank. The main difficulty is that, a smaller time-step is required to run the model (0.3 sec.), which proves extremely costly, compared to the 1.0 sec. used with the coarser mesh.

#### 5.3.2.2 Mesh Impact on the Solution and Time Step

Numerical integration of the discharge flow in TELEMAC, at cross-sections across the flume indicates that both meshes enable an accurate calculation of the mass flow. This is an important verification because the finite element technique is not mass conservative by default.

Results obtained with the 62,300-element mesh (Figs. 5.22 to 5.25) using a mixing-length model are comparable to the FCF data and to Lavedrine's calculations (1997). The main features are certainly recognisable although not always accurately reproduced. With the finer mesh (130,290) an identical situation is produced: The layer-averaged nature of the codes dominates the results, and no improvement is visible, especially where strong vertical flows are know to occur on the banks but are not well predicted. This means that it will be difficult to obtain a perfect representation of the FCF flow using a hydrostatic-pressure code. It also implies that enhancing the grid will not significantly improve the numerical solutions, and that the code's inaccurate predictions are mostly the result of its constitutive models' assumption.

Regarding the time step, time steps of 1.0 sec. were used for the coarse mesh, whereas a smaller value of 0.3 sec. was required for the finer grid to run satisfactorily. These time increments were used with the mixing-length model, however it appeared that smaller values would have been required with the $k$-$\varepsilon$ model, as discussed below. In the following simulations 600 seconds of simulations were run.

5.3.2.3 Turbulence Issues

With the $k$-$\varepsilon$ model, the mesh TELEMAC FCF-1 described above could not be used successfully. A first attempt to lower the number of vertical levels to 5 was used unsuccessfully (The programme kept failing after a few initial iterations). It was therefore decided to employ a mesh representing the lower half of the channel between bends 4 and 8, and maintain the node resolution as described before. This did not work either.

In general the $k$-$\varepsilon$ turbulence model was found to be very demanding of computer power, and could only be used with fewer nodes and much smaller time steps (0.05 sec. for the FCF), although not satisfactorily. The author understands that the HR Wallingford simulations could also have faced similar limitations (Lavedrine 1996, 1997). This would explain the absence of reference of the TELEMAC $k$-$\varepsilon$ model in Lavedrine's reports.

Since turbulence transport appeared to be negligible in the studied channel (see Section 5.4) and that lowering the time step would make a simulation with TELEMAC as costly as a fully three-dimensional one with CFX, the idea of using the $k$-$\varepsilon$ turbulence model was abandoned. The use of the mixing length model and the layered nature of the code implied that the bed shear stress was not calculated with TELEMAC (see section 4.4.3.2).

### 5.3.3 Numerical Discretization

It was decided to treat the problem in a simple and efficient manner. Consequently the default recommended set-ups were used for the treatment of the velocity and turbulence fields:

(i) Advection solution using by the Method of Characteristics (MOC) for the velocities;

(ii) Advection solution using the SUPG for the advection of the water depth;

(iii) Diffusion using the variational finite element method.

### 5.3.4 Numerical Issues and Convergence

Sub-iterations are implemented for the advection phase to attempt to obtain a reduction of four orders of magnitude on the convection and vertical velocities. The maximum number of iteration was set to 200 to try and reach this accuracy per time iteration, but is not needed. 100 iterations are set up as a default value for the calculation of the vertical velocity, but are not reached either. This indicated that the time-step second chosen by the author is adequate for example. Finally, a maximum of 60 iterations is authorised for the diffusion of velocities.

To take into account the non-linearities in the advection terms two sub-iterations are implemented, so that three internal iterations that make use of the latest convection velocities (instead of the previous time step values) are conducted. This is to help enhance mass-conservation

As mentioned in Chapter 4 there is no steady stage in TELEMAC, so the discharge is monitored until the simulation appears to be steady, which occurred before the end of the 600 1-second time steps implemented for the FCF model presented hereafter. This calculation was completed in about 15.5 hours of CPU time on a Sun Station Ultra 10/433 with 384 MB. This is the main benefit of the relatively "simpler" formulation of the three-dimensional problem. It constitutes an important criterion with regard to an industrial application of the code.

### 5.3.5 Initial Conditions

Initial conditions were set up for the water surface elevation and velocity terms. The water surface elevation was positioned to match a uniform flow profile and the unknown velocity field was set to zero.

### 5.3.6 Boundary Conditions and Sensitivity Analysis

5.3.6.1 Inlet and Outlet Boundaries

The boundary conditions are set up as described in Chapter 4 (section 4.4.3). A flow of 0.25 $m^3/s$ is implemented via a velocity profile at the inlet assuming a parabolic distribution across the channel width, and a mass-flow condition is imposed at the outlet. Assumptions for the turbulence quantities at the boundaries have default set-ups also described in Chapter 4. Referring to Alfrink and van Rijn (1983), such fixed conditions for the inlet should not be detrimental to the results, especially as the full-length FCF flume is modelled here, and the control over the numerical boundary conditions is at least as good as in the flume experiments. Consequently, no sensitivity analysis to the turbulence set-up, and velocity quantities at the inlet is presented.

5.3.6.2 Wall Roughness and Free Surface

As the mixing-length model is used, equation (4.54) is implemented at the walls. In TELEMAC roughness is usually implemented via a Chézy coefficient or a roughness height value turned into a Chézy coefficient using Ramette's formula (Janin et al. 1997).

The main impacts of adjusting the friction factor are an increase of the velocities at the walls and a modification of the free surface slope. The friction factor is expected to have a significant impact on the free-surface slope calculated in TELEMAC.

With the first model, a Chézy C of 63 is used (as in Lavedrine, 1996), which corresponds to a roughness height of 0.8 mm, calculated as an example in Chapter 4 for the FCF. With such a value the free-surface slope is found to be $1.11 \times 10^{-3}$, which represents an error of 12% with respect to the uniform-flow slope of $9.96 \times 10^{-4}$. This is equivalent to a height difference of 5.5 mm in the upper part of the channel.

A second simulation is conducted with the mixing length model, using a Chézy C of 74 to make the channel smoother. This value corresponds to a roughness height of 0.2 mm. The depth- averaged velocity maximum appears to be marginally faster, but little variation is observed at any of the four monitored cross-sections compared with the previous. Velocity increases by 2 cm/s in regions close to the walls and variation in the angle extremes is limited to ± 3 degrees at cross-section 5. The averaged free-surface slope is found equal to $9.79\times10^{-4}$, and the upstream water surface elevation underestimated by about 1.0 mm.

Another simulation is run for C = 68, corresponding to a roughness height of 0.4 mm. No significant variations in the velocity magnitude and direction are observed. The averaged free-surface slope is calculated as $1.01\times10^{-3}$, which entails a difference close to 0.5 mm at the upstream end of the full-length FCF flume (Fig. 5.17).

This confirms that it is perfectly feasible to adjust the model to obtain the correct "averaged" surface slope and demonstrates the equivalence between experimental and numerical roughness values for such a simple experiment (see Sections 4.3.3.4 and 4.4.2.2). What also proves interesting is the model's ability to replicate the expansion and contraction effects on the free surface position in the region of the cross-over (Fig. 5.18). For example, the flow expansion results in an increase of 2 to 3 mm of the flow, and the contraction is a decrease of 5 to 6 mm (Fig. 5.18 for C = 68), which is similar to FCF free surface maps presented by laboratory investigators (Hardwick, 1990; Lorena, 1992).

### 5.3.7 FCF Validation

#### 5.3.7.1 Velocity Field

Fig. 5.19 shows a depth-averaged plot of the velocity field obtained using a standard mixing-length model. Past the bend, part of the inbank water is ejected onto the outer bank where it travels in a skewed direction. The other part is slowed down against the inner bank where it is "rotated". The outer bank flow is ejected before reaching the middle of the cross-over, as the increase in the velocity in this region tends to show. The inner bank flow is further slowed as the flood plain water expands in the main channel. At the bend, it is also clear that the flow along the inner bank is faster that along the outer bank. These remarks about the general flow features in two dimensions indicates the model's potential to reproduce the flow pattern in a meandering compound channel with overbank flow.

More detailed outputs from the calibrated model are shown on Figs. 5.20 to 5.21, to be compared with Figs. 5.4 and 5.5. Figs. 5.20 and 5.21 are graphical interpretations of the outputs produced by TELEMAC, Fig. 5.22 to 5.25, which lack clarity and display localised spurious vertical velocities. On Fig. 5.20 cross-section 1, an isovel of velocity of about 40 cm/s is visible on the left bank and in the upper part of the overbank flow. Between this isovel and the channel centre, there is an isovel of 32-35 cm/s, followed on the right hand side by a velocity field around 30 cm/s. This is comparable to the FCF data set, although it under-predicts the magnitude in velocities. A similar comment can be made for cross-section 3, where the core of the velocity around 40 cm/s is located along the left bank, and lower 33-cm/s and 31-cm/s isovels are located in the centre and right part of the main channel. The error in the velocity magnitude ranges between 5 to 15% in places for these two cross-sections, but the overall flow pattern seems to be adequately reproduced. The velocity field at cross-section 5 seems quite uniform in the main channel at around 33 cm/s, with the exception of the right bank where a faster flow (40 cm/s) is calculated, which does not compare well with the data. It could be due to the strong velocity gradient and to an inadequate calculation of the vertical velocity component by TELEMAC. In the upper part of the channel, the velocity field is slightly faster than expected (40 cm/s) especially along the left bank. What is visible is also that there is a region of lower velocity between this isovel and the bank. At cross-section 8, the rotational structure is visible, although the layered nature of the TELEMAC calculation makes this less clear in the main channel. There is a central bulge of velocities around 30 cm/s starting on the upper right part of the channel, similar to the shape seen on FCF data plots. A very strong upward velocity on the left bank confirms the rotational nature of the flow, although it results from an inadequate calculation of the vertical velocity component. This surely affects the quality of the numerical solution, and justifies why a calculation of the bed shear stress based on a layered approach and on the local velocity could not be used. The inflow of flood plain water is visible in the upper left hand side of the figure with an intrusion of fast flow (40 to 35 cm/s) nearly until the middle of the main channel. The sharp velocity gradient predicted on the right bank also seems correct compared to Fig. 5.4.

At cross-section 1 (Fig. 5.21) the angle plot shows a quasi-horizontal gradient of the angle starting at 23 degrees on the right flood plain and progressively reversing to −12 degrees at the bed and to −19 degrees, in the right bottom corner. Although similar to the data this progressive grading does not quite reproduce the angle calculated during the FCF experiments above bankfull on the right bank (Fig. 5.5). A horizontal gradient similar to that of cross-section 1 is also visible at cross-section 3. Isocontours of up to −10 degrees are calculated on the bed and account for the helical component of the flow being deviated to the right. As this component rises in the water column it is progressively reversed to the left, with the 0-degree mark being located under the bank level at half level in the main channel. The sharp change in the angles, above bankfull at the main channel/flood plain interface indicates that there is little interaction between inbank and flood plain flow. This compares well with the data despite a slight under-prediction of the angle values. At cross-section 5, the vertical gradient of the angles appears very clearly, with a separation around the −16-degree angle line (vertical "dent"). Such a vertical representation was expected to be difficult to produce since the model is a layer-averaged. The formation of the next helicoid is also visible in the bottom left corner and the angle magnitude is well reproduced. The sharp angle gradient is also relatively good on both sides of the flood plain/main channel interface. Finally, at cross-section 8, the quasi-horizontal gradient of the angle is adequately reproduced over the water depth, and so is the sharp change in direction on the right bank. The flow mechanism is not only perfectly reproduced here, but so are the calculated angle values.

5.3.7.2 Secondary Currents

Secondary velocities for this series of models were calculated with TELEMAC over the entire section, and in more detail in the middle of the main channel. These velocities are reproduced on Figs. 5.26 to 5.29. The magnitude of the recirculation cell at cross-section 1 (Fig. 5.26) is underestimated by a factor of 2 to 3. The level of error is reduced at cross-section 3 (Fig. 5.27), but it remains large with a difference of 30 to 50%. However, because of the small magnitude of secondary currents in these areas, a small numerical error comparatively generates a large difference between the model and the data sets. At cross-section 5 (Fig. 5.28) the "non-existence" of the secondary current is clear. The beginning of the flow reversal is happening. The strong secondary recirculation at cross-section 8 (Fig. 5.29) is well reproduced. It varies between −14.0 cm/s (at 180 mm) and 12.0 cm/s, which compares with the range of values calculated from the FCF data, although it is slightly weaker close to the free-surface.

### 5.3.8 Discussion

In general, TELEMAC offers little flexibility regarding the formulation of the problem. It has been designed for fluvial and coastal applications only. This is particularly true of the way the mesh generator operates or the turbulence models and boundary conditions are formulated. This is why the code appears limited with regard to the present detailed investigation. TELEMAC is a large-scale problem code, primarily designed to investigate two dimensional flow features, possibly with layer variations due to density or heat variations. It is not properly speaking a three-dimensional code and therefore fails where strong vertical accelerations exist. One should expect such insufficiencies, especially when investigating a detailed laboratory experiment, and users should consider real-case scenarios for the code has been designed as better guidelines.

A general positive comment should be made with respect to the quality of the overall result offered by TELEMAC however. Despite the limitations of the formulation and the simplicity of the physical models, the general flow pattern is well reproduced, and, most importantly, at low computing cost. This implies that the flow in meandering two-stage channels is probably dominated by "simpler" mechanisms than in straight cases, and that the geometry "drives" the three-dimensional flow main features. As will be shown in the next section, TELEMAC outputs regarding the main flow directions are reasonably good compared with CFX. The main difference between TELEMAC and CFX lies in the degree of accuracy that can be expected from the models in three dimensions.

## 5.4 MODEL OF THE FCF USING CFX

### 5.4.1 Preliminary Comments

The aim of this section is to describe the computer models of the FCF constructed using CFX. More attention is given to the verification and validation of this code, since it does not have a track record of application to open-channel. Verification includes an investigation of discretization issues, such as the choice of the numerical scheme, mesh independence, the impact of round-off errors and boundary conditions. The first two issues deal with the numerical error occurring in the solution as a result of the model's construction, whereas, the third aspect is a more fundamental problem related to the sensitivity of the integral solution to its boundary values. This could be used to validate the work of Alfrink and van Rijn (1983) regarding the sensitivity of a flow over a trench to the inlet velocity fields. The model is then validated by comparison with the laboratory data, as was TELEMAC. Finally the impact of turbulence anisotropy and turbulence transport in the solution of meandering-two-stage-channel flows is discussed, based on the FCF results, and on a test for a straight compound channel flow reported in Cokljat (1993), after Tominaga *et al.* (1989). This is another useful part of the current research, as the comparison will inform on the degree of turbulence closure necessary to obtain satisfactory simulations of overbank flows in meandering channels.

### 5.4.2 Spatial Discretization

5.4.2.1 Grid Construction

Four numerical grids with different grid densities are constructed and used to determine the level of grid refinement necessary to achieve grid-independence for the FCF problem. They are referred to as grids CFX FCF-1, FCF-2, FCF-3 and FCF-4. The first grid (CFX FCF-1) consists of 49,200 cells. In the first stage of refinement this is modified to improve the cross-sectional resolution (CFX FCF-2, 109,224 cells). Following this further refinement is undertaken both in the cross-sectional plane and in the downstream direction to maintain a reasonable cell-aspect ratio (CFX FCF-3, 218,120 cells) and (CFX FCF-4 889,712 cells).

These grids are built following the description and formulas given in section 4.3.1.2. A multiblock approach is used as shown in Fig. 5.30. Details about how the different element sizes were chosen are given in Table 5.1. The reader is referred to section 4.3.1.2 for the terminology. Two of the grids, called CFX FCF-1 and CFX FCF-3, are shown on Figs. 5.31 and 5.32, for the exit cross-section.

| ***Location – Direction*** | **Mesh CFX FCF-1** | **Mesh CFX FCF-2** | **Mesh CFX FCF-3** | **Mesh CFX FCF-4** |
|---|---|---|---|---|
| ***Flood plain – Lateral*** *(on either side)* | 12 elements *regular* | 12 elements $r$ = 10 (MC) | 15 elements $r$ = 10 (MC) | 24 elements $r$ = 10 (MC) |
| ***Flood plain – Vertical*** *(above bank level)* | 4 elements $r$ = 1.5 (BL) | 5 elements $r$ = 2.0 (BL) | 7 elements $r$ = 2.0 (BL) | 11 elements $r$ = 2.0 (BL) |
| ***Main Channel – Lateral*** | 12 elements $r$ = 1.5 (*) | 28 elements $r$ = 4.0 (*) | 40 elements $r$ = 4.0 (*) | 64 elements $r$ = 4.0 (*) |
| ***Main Channel – Vertical*** *(in main channel only)* | 13 elements $r$ = 1.5 (*) | 15 elements $r$ = 2.0 (*) | 21 elements $r$ = 2.0 (*) | 34 elements $r$ = 3.0 (*) |
| ***Main Channel – Longitudinal*** *(with effects on all domain)* | 164 elements *regular along main channel* | 164 elements *regular along main channel* | 164 elements *regular along main channel* | 264 elements *regular along main channel* |
| ***Total No. of Cells*** | 49,200 | 109.224 | 218,120 | 899,712 |

(*) *indicates a bi-directional bias, see Chapter 4;* (BL) *from the bank level in the upward direction,* (MC) *from the main channel towards the flume sides.*

**TABLE 5.1 – CFX Grid Characteristics Used to Model the FCF Flume.**

5.4.2.2 Grid Refinement and Mesh Independence

The process of refinement shown in Table 5.1 follows the recommendation given in Roache (1989) and the AIAA (1998) to establish mesh independence. It is also designed to enhance the grid resolution in regions where sharp gradients of the variables are likely to occur, notably at the banks due to the interaction between overbank and inbank flows. Care is also taken regarding the position of the first nodes close to the walls to ensure a suitable transition between the law of the wall and the fully turbulent Navier-Stokes equations (Fig. 4.9).

As pressure plays an important role in the solution of such a fully three-dimensional problem it has been used to investigate grid independence by comparing values of the non-dimensionalised pressure, $C_p$, at the bed along the channel centreline (Younis, 2000). Fig. 5.33 shows the value of computed pressure coefficient, $C_p$, along the main channel centreline at the bed for the four densities of numerical grid. Fig. 5.33 indicates that the numerical error for $C_p$ between CFX FCF-1 and CFX FCF-2 is large (around 60%), whereas it is significantly reduced between CFX FCF-2 and CFX FCF-3 (around 5%).

The velocity field is obviously affected by the level of resolution and a typical error between the predictions at cross-section 8 with CFX FCF-1 or CFX FCF-3 is shown on Figs. 5.34 and 5.35. The main error in velocity is in the region close to the wall, which justifies the design improvement displayed in Table 5.1. Overall the difference in velocity is within 10% of the typical maximum velocity inside the domain and about 30% to 40% of the velocity in the immediate vicinity of the walls (Fig. 5.34). In terms of the flow direction at cross-section 8, there are larger errors at the walls, 10-15 degrees, but the error is also spread slightly in the domain (Fig. 5.35). This means that a coarse grid still captures the essence of the flow dynamics away from the walls. These differences are much smaller between CFX FCF-2 and CFX FCF-3: The difference in velocity is globally within 2% of the maximum velocity (Fig. 5.36), and a typical difference in angle is ±0.2 degrees (Fig. 5.37). One interesting feature on both figures is the fact that the difference seems to "follow" the flow features in the sense that there is a distinct resemblance between the error contours and the flow contour features (Figs. 5.34 to 5.37). In the case of CFX FCF-2 and CFX FCF-3, the grid enables a good representation of the velocity profile at the walls.

These comments mean that the solution obtained using CFX FCF-3 is relatively free of major numerical error induced by the grid, and that it is reasonable to assume that CFX FCF-2 represents an acceptable resolution threshold. This analysis shows that a high resolution seems to be required to alleviate the error due to spatial discretization at the walls.

Grid CFX FCF-4 could not be run at the University of Glasgow due to insufficient computing power, and was consequently used for a simulation on a larger workstation at University of Nottingham where it converged satisfactorily. However, machine compatibility problems have prohibited the result obtained being viewed at Glasgow University.

### 5.4.3 Numerical Discretization

Intitial models of the FCF flume by the authors used the hybrid discretization scheme. Although very attractive for its robustness it was apparent from preliminary results that the first order accuracy of the scheme did not resolve the sharp velocity gradient at the banks satisfactorily, even after considerable local refinement of the grid. It is therefore decided to adopt the third order accurate discretization scheme CCCT (Gaskell and Lau, 1988). The boundedness property of this scheme is essential for the calculation of turbulence quantities, particularly $k$. The level of resolution that is sought here justifies the adoption of such numerical scheme.

### 5.4.4 Solvers

It was mentioned in Chapter 4 that Stone's solver and the AMG would be used in the course of this work. Stone's method because it is an ADI-type method, i.e. a common approach in CFD work, and the multigrid solver because it is particularly efficient for complex problems (complex and fine grids). The use of multigrid acceleration methods has indeed been "wished" in the conclusions of recent publications for river flow problems (Sinha *et al.* 1996; Mesehle and Sotiropoulos, 2000).

In the present work it was found that AMG overshadowed the other solvers, in terms of time-efficiency but also in terms of reliability. In particular, as the mesh size decreased in the FCF problem to reach a resolution of 218,120 elements, Stone's solver faced severe convergence difficulties and required finer reduction and relaxation parameters than AMG (Table 5.2), which in turn made it more costly. In the case of the FCF, difficulties in reducing the smooth error[1] could explain the difficulties experienced in using Stone's solver. In natural channels however, these would add up to the relative lack of "grid order" born out of the irregular geometry, and to distorted grid elements, which would result in the deterioration of the performance of Stone's and ADI solvers. These findings support comments made in Sinha *et al.* (1996) and Meselhe and Sotiropoulos (2000) concerning the usefulness of multigrid methods in complex channel problems.

Multigrid solvers, and in particular the Algebraic Multi-Grid (AMG) solver used here, are desirable in complex open-channel configurations. They were found to be efficient and reliable compared to other methods such as the ADI or Stone's methods, especially with fine grids, and in the cases where the RSM turbulence model was implemented.

### 5.4.5 Convergence

Convergence is closely monitored for all simulations discussed here, by ensuring (i) a reduction of four orders of magnitude for all residuals and (ii) monitoring the stability of the variable values at the measured locations in the model after (i) has been achieved. These criteria have been illustrated in detail in Chapter 4, where the convergence histories that were presented were those obtained for grid CFX FCF-3.

### 5.4.6 Round-off Error

A numerical simulation using the FCF-2 grid was conducted using double precision arithmetic. Comparison of the results confirms that there was no significant difference between the numerical outputs of the single and double precision simulations. One noticeable impact is on the aspect of the residual convergence curve, which appears to be

[1] Errors which wavelength $\lambda >> h$, the element size, i.e. numerical errors which are difficult to eliminate on "fixed" grid algorithms and are likely to become more important as the element size decreases.

smoother. This suggests that double precision may be useful when the numerical error between successive iterations results in instability or lack of convergence, in particular during the initial stages of the simulation. There is no advantage in using double precision in the simulations reported here, and round-off error is not an issue.

### 5.4.7 Boundary Conditions and Sensitivity Analysis

Boundary conditions were determined as in Chapter 4 (Section 4.3.3). Because of the difficulties associated with the accurate definition of boundary values, and the significant impact that roughness is known to have in the determination of the position of the free surface it is necessary to evaluate the sensitivity of the numerical solution to variations in the values used.

#### 5.4.7.1 Inlet and Outlet Boundaries

The calculation of the inlet and outlet boundary conditions is as presented in Chapter 4 (section 4.3.3). A constant velocity and a logarithmic profile similar to equation (6.1) have been successfully implemented but lead to similar results in the meander region.

It is decided to test further the impact of the boundary conditions at the inlet and outlet. To do so a simulation was run using periodic boundary conditions. Periodic boundary conditions use the fields from the outlet as the next iteration inlet conditions, and virtually help model an infinite channel. Their use ensures fully developed flow in the model once the solution has converged. This is an important parameter to test the impact of the outlet boundary conditions such as equation (4.27) on the solution. Similarly it offers the possibility to test the impact of the inlet condition on the solution by enabling a comparison of the solutions obtained at sections 1, 3, 5 and 8 with a simple inlet profile and a "real" three-dimensional in-coming velocity profile modelled through the periodic boundary. Periodic boundary conditions are easily applied for the simulation of gravity-driven flow conditions in prismatic channels.

The results indicate that little variation seems visible at monitored cross-sections between the model using the above condition and the model with standard inlet/outlet (I/O) types of boundaries (Figs. 5.38 and 5.39). This conclusion is important as it validates the choice of an inlet/outlet boundary condition and ensures that the latter will not affect the solution significantly. This confirms the conclusion of Alfrink and van Rijn (1983) who suggest that, for flow over a trench, inlet boundary condition accuracy is of *"minor importance"*, especially when the inlet is situated sufficiently far upstream from the study area. Stansby and Zhou (1998) also implemented such simple conditions in their study of flow over a trench. It is encouraging for practising engineers, as it appears that coarse or synthetic inlet conditions might be satisfactory to carry out CFD simulations of river flows, for which detailed field data are usually scarce. Such conditions are also the more "natural" to river engineers who have used discharge values in the numerical modelling of river flow ever since the implementation of the first one-dimensional models.

In the present study independent upstream and downstream boundary conditions are therefore implemented with confidence. This technique has also been chosen as it is the most convenient approach when simulating a river, where the geometry of the inflow and outflow boundaries will be different and complex, and therefore not suitable for periodic boundaries, and where gravity might not be sufficient to drive the flow. Additionally, periodic boundary conditions are more computer-intensive and prone to instability, in particular if used to initiate a simulation (without prior calculation). In the FCF model, such a difficulty resulted in a 50% increase in CPU time. There is therefore no benefit in using such conditions here.

5.4.7.2 Wall Roughness and Pressure

Wall roughness and the pressure term on the lid interact, and it is necessary to adjust these in parallel during the model validation. As mentioned previously a wall function (Table 4.1) has been implemented to model the effect of bed roughness on the velocity field. With such an approach, the only variable that requires to be estimated is the roughness height value. This estimation corresponds to the usual calibration work carried out by modellers when modelling river flows using one-dimensional and two-dimensional models. Once $k_s$ values have been estimated (see section 4.3.3.4) it is necessary to check their impact on the solution, and in particular on the calculated water surface profile computed as a pressure head on the rigid lid. This means that only small pressure variations other than the head loss due to the slope should be present on the lid. It is an important verification from a numerical point of view as well, as excessive pressure can impact on mass distribution and mass conservation, and affect the validity of the numerical solution.

Assuming a quasi-uniform flow, the slope of the free surface should be close to the bed slope. Consequently, for the FCF simulations the pressure gradient along the channel centre line should be equal to 9.96 Pa/m to be representative of the uniform flow free surface. Comparisons using different roughness values were conducted as shown on Fig. 5.40. The pressure gradient varies between 12.4 Pa/m ($k_s$ = 0.8 mm) and 9.9 Pa/m ($k_s$ = 0.2 mm). This means that for values larger than $k_s$ = 0.2 mm the wall surface is too rough and would result in a biased pressure distribution with regard to the validation data representing experiment B23, however, no large variation of the velocity and turbulence fields is noticeable for the different $k_s$ values. With a coarser roughness the flow appears to travel marginally slower, especially close to the walls (compare Figs. 5.42 to 5.45 with 5.46 to 5.49). This is most probably because the chosen roughness values correspond to relatively smooth conditions. It is expected however that roughness will affect the velocity field in the simulation of turbulent river flows.

The roughness values calculated in Chapter 4 seem to work rather well for the model of the FCF. This is probably due to the nature of the model however, a flume which presents smooth regular walls, mostly incorporating skin roughness effects. In addition the level of discretization at the walls is also very fine and enables a good numerical representation of the momentum losses.

### 5.4.8 Validation

5.4.8.1 Velocity Field

Fig., 5.41 shows surface velocity vectors from the FCF model. These should be compared with experimental pictures such as those shown in the paper by Shiono and Muto (1998) for a similar depth ratio. Qualitatively the results seem good, in particular the direction of travel following the interaction between main channel and flood plain in the centre of the flume and the deceleration/acceleration of the flow as it crosses the main channel. A more detailed analysis is conducted hereafter at cross-sections 1, 3, 5 and 8 for $k_s$ = 0.8 mm (Figs. 5.42 to 5.45) and 0.2 mm (Figs. 5.46 to 5.49).

Figs. 5.46 shows the results obtained with the $k$-$\varepsilon$ model with $k_s$ = 0.2 mm, and the same set up is used in the following paragraphs. The velocity field at cross-section 1 is well reproduced with the maximum isovel (40 cm/s) located along the left bank and connected to the in-coming flow from the upstream flood plain. The 35-cm/s isovel however is not well positioned, which indicates here that the flow in the channel is marginally slower (about 10%) than in the observations. The flow pattern is also well reproduced at cross-section 3 at the bend, although the velocity field is again underestimated by 10%. At cross-section 5, The centre of the 40-cm/s isovel is correctly centred around 600mm, and the overall velocity pattern is adequately calculated. The velocity distribution is very close to the original data set for this section, even if the 40-cm/s isovel remains too narrow. The vertical gradient of the velocity on the top right bank, as the water contracts to jump on the flood plain, also seems to be correct. Cross-section 8 is very well modelled. The water coming from the flood plain is gradually slowed down with depth, in the horizontal plane. The shape of the isovel in the top part of the main channel indicates a rotating flow. The velocity intensity in the central region is correctly reproduced (30.0 cm/s). Strong vertical velocity fields arising from the fluid rotating are also present on each bank walls (35 cm/s on the left and 40 cm/s on the right bank). The velocity distribution at cross-sections 5 and 8 is more accurate than at 1 and 3, although the general velocity pattern is reasonably well modelled and the error in magnitude is within 10 %.

The direction of the flow is given by the angle deviation from the normal to the cross-sectional plane. The results obtained with the current simulation are shown on Fig. 5.47. At cross-section 1, inside the main channel and below mid-depth, the angles are well reproduced showing a quasi-horizontal distribution of the angle gradient starting from −10 degrees at the bed. However, as it shifts upward, towards the channel interface, a difference between model and the measured data appears. The maximum angle calculated from the oncoming flood plain flow on the right bank is only 22.5 degrees compared with 45.0 from the measured data, with both $k$-$\varepsilon$ model and RSM. This means the flow is travelling parallel to the flood plain walls in the model whereas the data would indicate that it should be at an angle of about 20 degrees. This error in the model could be due to the fact that the flood plain velocity field is this area of the model is still dependent on its boundary values set upstream in a "straight" direction. On the other hand, the TELEMAC full-scale model produced identical results in this region, which then leads to question the quality of the laboratory measurements in this region.

While the inbank angle pattern and magnitude are very well calculated for the inbank flow at cross-section 3, a discrepancy of a few degrees is visible above the bank level, in the main channel and on the left bank. It remains a small difference however (between 5 and 7.5 degrees). Cross-section 5 is very well reproduced. The vertical line of –15 degrees is correctly calculated, and the difference at the bed could in fact arise from the post-processor which attempts to re-attach the iso-angle line in an inappropriate manner. The positive angle cell along the left bank, although slightly too large, is well accounted for, as are the angle gradients on both banks. Finally the excellent results obtained at cross-section 8 confirm that this model reproduces very well the overall flow pattern along the channel meander.

Combining both velocity direction and intensity, the model results are plotted as vectors on Figs. 5.50 and 5.51. These plots show what is happening in the water column as a result of the interaction between overbank and inbank flows, and illustrate how well the model output compares with the data set. This is further confirmed by numerical tracer experiments (Fig. 5.52), compared with the laboratory data of Willetts and Hardwick (1993) shown on Fig. 5.53 and some FCF pictures provided by Professor Ervine, Fig. 5.55. It is clear that the velocity field is well reproduced by the current model.

The simulation that was described so far makes use the $k$-$\varepsilon$ model. A second simulation was run using the RSM, but it was found to be relatively unsuccessful (Figs. 5.48 and 5.49). Firstly, it did not improve the solution obtained with the isotropic $k$-$\varepsilon$ model, and secondly it required a longer simulation time to converge (29% extra time). This raises the question of the nature of turbulence transport in the FCF Series B, especially as a third, simpler turbulence model was used with reasonable success in TELEMAC. This would suggest that turbulence transport is secondary to the cross-flow effects, by far, in creating the flow pattern observed here. Further answers are provided in two specific sections on turbulence presented hereafter (sections 5.4.8.3 and 5.4.9).

5.4.8.2 Secondary Currents

Using the above data it is possible to derive secondary current terms in the plane perpendicular to the main channel centreline. These secondary currents are expected to compare reasonably well with data because the velocity field is accurately reproduced. The main re-circulation cell at cross-section 1 and 3 are reasonably well predicted, although the velocity is lower at the bed than one would have expected from the measured data, Figs. 5.55 and 5.56. This could be due to inaccurate simulation at the walls in the numerical model or equally to inaccurate measurements close to the wall as previously underlined for cross-section 1. The results obtained at cross-section 5 reflect the vertical plunging effect and the quasi-absence of lateral circulation, Fig. 5.57. The reversal of the helical motion begins at this point, where the model predicts secondary current values between –1.5 cm/s and 5.0 cm/s, compared with –2.5 cm/s to 5.0 cm/s measured in the FCF. The predictions obtained for cross-section 8 are good, Fig. 5.59. The magnitude of observed velocities in the main re-circulation cell are between –22.0 cm/s and 14.0 cm/s and super-impose with the FCF data. These correspond to the strongest re-circulation observed in the experiment.

From a geomorphological viewpoint this is very important since these structures are believed to be crucial in fluvial hydraulics with regard to sediment transport and bank erosion. The existence of the helical motion observed in the laboratory (Fig. 5.3) is reproduced beyond doubt by CFX (Fig. 5.59). One can therefore be confident that such application could be extended to investigate shear stresses and erosion processes at the bed.

5.4.8.3 Turbulence Field

Considering the turbulence data provides an additional information regarding the physical validity and the quality of the processes that are reproduced by the code. It could also help understand the nature of the turbulence mechanisms arising in such complex flow and understand why the anisotropic turbulence model is unsuccessful. This would be an important knowledge to gain since the nature of turbulence should condition mixing and transport processes, and determine the type of turbulence model that is needed for the flow that is considered. Both the outputs of the $k$-$\varepsilon$ model and RSM are considered below at cross-sections 3 and 8b (Fig. 3.1).

At cross-sections 3 and 8b comparison between the $k$-$\varepsilon$ model and RSM are carried out between turbulence kinetic energy outputs. As can be seen on Fig. 3.60, there is little difference between the distribution of $k$ across cross-section 3, except for the fact that the outputs with RSM is about 15% lower in the centre of the structure. Away from the centre, the turbulence kinetic energy is about the same. At cross-section 8b both outputs are very similar (Fig. 5.61), although the $k$-$\varepsilon$ outputs are about 12.5% larger than with the RSM. General features regarding the distribution and size of $k$ across the half cross-section are that turbulence kinetic energy is two to three time larger at 8b than at 3, and centred around the upper middle part of the channel at the interface level. Unfortunately, no laboratory data was available for the turbulence kinetic energy to evaluate the quality of the simulations.

The values of the Reynolds stresses used in the following investigation are the solution of the additional transport equations in the RSM, and those calculated by the author from the knowledge of the computed velocity field and turbulent eddy viscosity with the $k$-$\varepsilon$ model, using the Boussinesq assumption (equation 2.4).

At cross-section 3, both computed $T_{yx}$ plots exhibit large Reynolds stresses along the left bank, and a concentric region around 0.10 N/m$^2$ in the channel centre (Fig. 5.62). None of the models seem to have been able to reproduce a smaller intermediate region around – 0.10 N/m$^2$ shown on the FCF data. Similarly, the region of very high shear located in the bottom left corner of the main channel is not reproduced by either of the turbulence models. Concerning $T_{zx}$ (Fig. 5.63) the concentric structure observed in the top right corner of the main channel at cross-section 3 (0.25 N/m$^2$) is reproduced by both turbulence models. Stress values around 0.20 N/m$^2$ are also well calculated along the right bank. However the quality of the laboratory data is very poor and it is difficult to draw firm conclusions.

At cross-section 8b, the numerical output (Figs. 5.64 and 5.65) seem simpler and much larger than the plot of the measured values. Both $k$-$\varepsilon$ and RSM yield quasi-identical distribution of $T_{yz}$ and $T_{zx}$ respectively although they are very different from the measurements. The large difference between data and computed values is difficult to explain because of the lower quality of the data at this location. Moreover, the similarity between the two models outputs accentuates the doubt concerning the quality of the data. However, the fact that either turbulence model yields the same answer would tend to indicate that there is little effect of anisotropy behind the flow mechanisms at the cross-over.

5.4.8.4 Bed Shear Stresses

Shear stresses were collected at section 3 during experiment FCF B23. This section is therefore used here for further validation of the CFX model. The calculation that was performed to obtain the bed shear stresses from the CFX results is based on the fact that in the vicinity of the walls the bed shear stress $T_b$ is equal to:

$$T_b = C_\mu \rho^2 k^2 \tag{5.2}$$

Fig. 5.66 shows a comparison of the models' outputs with the data. Although both turbulence models accurately capture the trend of the profile as well as the peak values with grid FCF-3, there is some discrepancy in the main channel. The error is of the order of 80% in places, but an average figure for the error would be about 20%, which is satisfactory considering the accuracy of bed shear stress measurements in the laboratory, and also that no additional calibration was performed to tune the model at the walls. A good point from a design point of view though, is that the extremes, located at the walls (banks), are well predicted. What is also significant is that there is little difference between the two turbulence models. This further confirms the secondary role that turbulence transport plays in such channel configuration for an overbank flow. Because of equation (5.2), the reasonable match between experimental and numerical data on Fig. 5.66 also implies that the turbulent kinetic energy plots shown on Fig. 5.60 are likely to be qualitatively representative of reality.

However, it should be noted that the calculation of the bed shear stresses has appeared to be dependent on the grid resolution at the walls, Fig. 5.66(a). Indeed the author found that with mesh FCF-1 for example calculated bed shear stress were significantly larger than with FCF-3 or FCF-2. With FCF-1 the error is five-fold, which shows the dependency of such calculation upon the mesh resolution when mesh-independence has not been established. The solution with FCF-2 is in general as good as that with FCF-3, and both could be used for design purpose. However, variations of 10 to 15% still exist between the outputs obtained with FCF-3 and FCF-2, although FCF-2 was deemed as mesh-independent regarding the velocity and pressure field. If such low resolution is necessary, this seriously compromises the immediate applicability of CFD to sediment entrainment.

Figs. 5.67 to 5.69 show the predicted shear stresses at sections 1, 5 and 8 using FCF-3 and the $k$-$\varepsilon$ turbulence model. Based on the previous discussion and further evidence provided by Knight (SERC, 1993, Vol. 7(A)), they are believed to be relatively accurate in their description of the stresses. They show that the largest stresses are found to be located:

(i) where the flood plain flow intersects with the main channel;

(ii) along the left bank where the helicoid is formed;

(iii) and also on the right flood plain where the main channel flow is ejected onto the flood plain (Sections 5 and 8b).

These characteristics are representative of overbank flows, and constitute a main difference with inbank flow features (SERC, 1993, Vol. 7(A)). In general the stress in the main channel is lower than for inbank cases, especially on the bottom corner of the right bank.

5.4.8.5 Partial Conclusion

The previous results have indicated the capacity of the $k$-$\varepsilon$ model to satisfactorily represent the velocity field in a meandering channel with an overbank flow. In particular, the results regarding secondary velocities and bed shear stress are good, and underline the model's ability to account for complex flow structures occurring in natural channels. Therefore, this tends to show that in such channels, where the flow features are mostly "geometry-driven", a simple turbulence model would be adequate to account for large secondary currents provided the level of discretization is sufficient.

The comparison of the numerical data with the FCF fully validates the model and demonstrates the ability of CFX to reproduce river-like fluid flows. The above simulation, using FCF-3 and the $k$-$\varepsilon$ turbulence model, required 45 hours of CPU time on a Sun Ultra10/433 with 384 MB RAM (about 58 hours with RSM). On the other hand the simulation using FCF-2 required about 15 hours, proving quite adapted and performing rather well compared to the TELEMAC model. This numerical solution did not appear to be sensitive to the turbulence model used, as both the $k$-$\varepsilon$ and RSM model previously described yielded the same answer. This is discussed in the next section.

### 5.4.9 Turbulence Anisotropy in Two-Stage Channels

5.4.9.1 Problem Statement

Two models of turbulence were used in the FCF simulation: the standard $k$-$\varepsilon$ model and the RSM of Launder, Reece and Rodi (LRR-RSM, 1975). However, the analysis of the FCF simulations carried out with CFX using different levels of turbulence closure, revealed that no improvement was to be gained with an increase in the level of the description of turbulence. This leads us to question the importance of turbulence transport in the Series B flows, especially as the results obtained with the two- and seven-equation models seem to compare well with the laboratory data set.

To test this idea a simulation has been conducted using the results of Cokljat (1993) for a straight compound channel based on the data presented by Tominaga *et al.* (1989). As previously reported Cokljat found that it was necessary to use a complete RSM model to reproduce all of the velocity characteristics in such a channel. So the simplified LRR-RSM used by the authors is not expected to perform as well as the full model. In any case the velocity pattern of the straight compound channel will not be reproduced with the $k$-$\varepsilon$ model, which is isotropic. The aim of this validation is to demonstrate that the simplified RSM manages to reproduce most of the turbulence-generated distortion of the stream-wise velocity at the interface between the main channel and the flood plain. The reasoning is that, should this be true, the identical solution provided by both $k$-$\varepsilon$ and RSM in the previous FCF Series-B test would suggest that turbulence anisotropy is not a significant factor in determining the velocity field in Series-B type flows. This would have implications for the simulation of flood flows in natural rivers.

### 5.4.9.2 Numerical Verification

A slightly different approach is adopted here to evaluate the grid-dependency of the numerical solution of Tominaga's model. Because the author is only interested in the streamwise velocity at a localised cross-section, it is as easy to compare the entire solution for this with two grids of different density, CFX T-1 and CFX T-2. T-1 is made of 54,000 elements and T-2 109,200. In terms of resolution, they compare with CFX FCF-2 and CFX FCF-3 respectively. As can be seen from Fig. 5.70, there is little difference between the two solutions. Mesh independence can therefore be assumed

Regarding the testing of the boundary conditions on the other hand, lack of data for the slope and the inlet velocity field in Tominaga's experiment (Cokljat, 1993) restrict such verification for the straight channel. However, a smooth flow condition is assumed, and it is expected that this should reduce the impact of the wall boundaries on the velocity and turbulence fields.

### 5.4.9.3 Validation and Conclusions

Both $k$-$\varepsilon$ and the simplified LRR-RSM are implemented, and the outputs for the streamwise velocity are shown on Fig. 5.71. What can be seen is that the second-order closure model produces a clear division between inbank and overbank flows, with the isovel at the interface slightly distorted towards the channel inside as indicated by the arrow. The position of the isovels across the section is correct and, in general, this model compares well with Cokljat's results shown on Fig. 5.72. A comparison of the non-dimensional velocity field carried out for two flow conditions (0.029 and 0.057 $m^3/s$, and Re = $3.5\times10^4$ and $7.0\times10^4$) using LRR-RSM, Fig. 5.73, also proves that the Reynolds number does not alter the solution, and that the conclusions drawn from a lower Reynolds number test will be equally valid in higher Reynolds number conditions, such as that of the FCF experiment B23. The $k$-$\varepsilon$ model presents a more uniform distribution of the velocity field, and a faster, non-physical flow on the flood plain. The isovels at the interface are not distorted towards the inside. This implies that even if the simplified LRR-RSM does not produce quite enough distortion to match the data it still produces substantially more anisotropy than the $k$-$\varepsilon$ model, which supports the argument that turbulence transport is not a dominant phenomenon in the Series-B flows for which both models produce very similar results. This would also explain why a simple mixing-length model such as that used in TELEMAC was sufficient to obtain a reasonable comparison with data.

### 5.4.10 Discussion

The FCF results are encouraging since complex secondary motions are believed to be important in influencing channel morphology as well as the transport and mixing of suspended quantities. Being able to model such complex flow features at low cost could enable better understanding of the mechanics of sediment entrainment, and help enhance the current sediment transport models. In the future, such codes could hope to be related to dynamic bank and bed erosion models. At a lower level, the outcome of this study also shows that it is possible to apply CFD codes such as CFX to river hydraulics problem, when detailed velocity fields are required.

The outcome of this work also conveys the idea that turbulence transport could be of minor importance for engineering applications in meandering compound channels, where the horizontal shear effects between two flows travelling in different directions are much more important. This conclusion is important for practitioners because, it suggests that for natural channels, a simple turbulence model is sufficient to reproduce the flood velocity field reasonably accurately and at relatively low cost. It also implies that turbulence anisotropy plays little role in generating the velocity field in meandering compound channels, which would be a useful conclusion regarding the physics of such flows.

This work made use of an Algebraic Multi-Grid (AMG) solver and confirmed, for the first time to the author's knowledge, the expectations of other researchers (Sinha *et al.*, 1996; Mesehle and Sotiropoulos, 2000) regarding its efficiency and particular adequacy to treat open-channel geometry problems. This solver proved reliable at all times without requiring excessive under-relaxation, even when the RSM was implemented.

This strengthens the idea that CFD can be accurately applied for predictive river engineering provided a rigorous numerical treatment of the modelled flow is done, as indicated in the verification part of this paper. This section could constitute the verification framework for a wider application of CFD to such river flow problems.

A summary of all the tests conducted by the authors is presented in Table 5.2.

| | GRIDS | FCF-1 | FCF-2 | FCF-3 | FCF-4 | T-1 | T-2 |
|---|---|---|---|---|---|---|---|
| | No. of Elements | 49,200 | 109,224 | 218,120 | 899,710 | 54,000 | 109,200 |
| | **TESTS/MEANS** | | | | | | |
| **Num. Accuracy** | Mesh Independence Solution Test | | ✓ | ✓ | ✓ | ✓ | ✓ |
| | Double Precision Run | | ✓ | | | | |
| **Boundary Conditions (BC)** | Law of the Wall (Position of first nodes/y+) | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| | Wall BC ("calibration" roughness and pressure) | | ✓ | ✓ | | | |
| | Impact of Inlet/Outlet BC (developed flow) | | ✓ | | | | |
| **Discretization** | Hybrid | ✓ | ✓ | | | | |
| | QUICK with Smart Limiter Function | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| **Solver** | Stone | ✓ | ✓ | difficult | | | |
| | Multigrid | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| **Validation** | Good Comparison with Data | | ✓ | ✓ | ✓ | ✓ | ✓ |
| **Turbulence Nature** | k-ε | ✓ | ✓ | ✓ | ✓ | ✓ | |
| | RSM | | ✓ | ✓ | | ✓ | ✓ |

**TABLE 5.2 – List of Tests conducted with CFX for the FCF Model**

## 5.5 SUMMARY AND CONCLUSIONS

This chapter has demonstrated the ability of commercially available CFD codes to model complex velocity fields in compound meandering open-channels. In doing so it has also proposed a methodological approach to verify the mathematical validity of the model and demonstrated that general CFD techniques have the potential to be used in engineering practice.

A practical finding of this work is that, in meandering channels with overbank flow a proper discretization of the problem seems more important than the choice of turbulence model. In fact, turbulence anisotropy has been found to have little impact on the quality of the numerical solution, which suggests the conclusion that turbulence transport might play little role in determining the velocity field in meandering compound channels. This is important to practitioners, as the complexity of turbulence models is a source of numerical difficulties and extra cost, but also to theoreticians interested in the mechanics of such flow. It also confirms early comments made by experimentalists Sellin *et al.* (1992) regarding the FCF Series B programme: "*The co-flowing lateral shear stress (zero mass transfer), so influential in the straight channel case, is insignificant [...] here. The mechanisms arising from the cross-flow and driven by the horizontal shear layer are much more important [...].*". Equally important is the fact that, although spatial discretization matters to obtain a good solution throughout the entire domain until the boundaries, coarse grids seem to be able to capture the main flow features away from the walls. This of course limits the quality of the solution and its usefulness to predict erosion and other dynamics effects at the walls, but it signals the possibility to analyse complex flows in much larger domain with a reasonable accuracy.

The CFX FCF model has also shown some very good behaviour regarding the modelling of the free surface when the rigid lid is located along the mean free surface slope. This is interesting, because it indicates that provided these fluctuations are small, a rigid lid model is perfectly adequate for engineering applications. Moreover, sensitivity analysis has shown that CFD models could in fact be as easily calibrated in a similar way to other open-channel flow models. A further improvement would be to relax the rigid lid position using these pressure variations. This work has also confirmed the relative lack of sensitivity of rigid-lid models to the inlet conditions. In a way, this is interesting for the industry, as the collection of detailed velocity fields is rarely possible and has to be substituted by bulk flow values (see Chapter 6). It has also illustrated the potential of multigrid solvers claimed in recent research. The free-surface code TELEMAC has also produced good results regarding both the free surface variations and the general flow pattern, in particular in the regions of strong horizontal shear layer. Although the free surface variations were expected to be a major attribute, limitations on the level of discretization, the treatment of pressure and turbulence model were initially considered as serious threats to the code's ability to handle such complex flow. The level of detail available in TELEMAC is lower compared to that in CFX and the "layer" approach dominates the solution (which justifies good comparisons where the horizontal shear effects are dominant). The calculation of the accurate vertical velocity components and pressure has appeared to be badly missing, e.g. in the region of strong vertical acceleration at the walls at cross-sections 1, 5 and 8, Figs. 5.22, 5.24 and 5.25, as also recently reported in Stansby and Zhou (1998). This paper further implies that a full calculation of pressure is more important than the accuracy of the turbulence closure, for the calculation of a flow over a trench.

The next phase of this research programme is to model meandering reaches of the Rivers Severn and Ribble. Predictions from these models will be compared against detailed field measurements collected during the winters 1998-1999 and 1999-2000.

## 5.6 REFERENCES FOR CHAPTER 5

1. AIAA (1998) "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations", AIAA Guide G-077-1998, AIAA.
2. Alfrink, B.J., van Rijn, L.C., (1983) "Two-Equation Turbulence Model for Flow in Trenches", J. Hydr. Eng., Vol. 109, No. 3, pp. 941-958.
3. Cokjlat, D.,(1993), "Turbulence Models for Non-circular Ducts and Channels", PhD Thesis, City University, London, UK.

4. Ervine, D.A., Sellin, R.J., Willetts, B.B., (1994), "Large Flow Structures in meandering Compound Channels", Proc. 2nd Int. Conf. River flood Hydraulics, York, UK. pp.
5. Gaskell, P.H., Lau, A.K.C., (1988) "Curvature Compensated Convective Transport: SMART, a New Boundedness Preserving Transport Algorithm", Int. J. Num. Meth. In Fluids, Vol. 8, pp. 617-641.
6. Hardwick, R. (1990), "Measurements of Local Surface Levels in the S.E.R.C. Series B", Technical Report, Civil Engineering Dept., University of Aberdeen.
7. Janin, J.M., Marcos, F., Denot, T., (1997), "Code TELEMAC-3D Version 2.2 – Note Théorique", Report HE-42/97/049/B, EDF-DER, LNH Chatou, France (In French).
8. Lane, S.N., Bradbrook, K.F., Richards, K.S., Biron, P.A., Roy, A.G., (1999), "The Application of Computational Fluid Dynamics to Natural River Channels: Three-Dimensional versus Two-Dimensional Approaches", Geomorphology, Vol. 29, pp. 1-20.
9. Lavedrine, I., (1997), "Evaluation of 3D Models to River Flood Problems", Report TR26, HR Wallingford, UK.
10. Lavedrine, I., (1996), "Evaluation of 3D Models for Flood Applications", Report TR6, HR Wallingford, UK.
11. Lorena, M.L. De Lima Da Silveira, (1992), " Meandering Compound Flow", PhD Thesis, Civil Engineering Dept., The University of Glasgow.
12. SERC, (1993), SERC FCF Series B Report, Vol. 1 to 8.
13. Roache, P.J., (1989), "Need for Control of Numerical Accuracy", J. Spacecraft and Rockets, Vol. 27, No. 2, pp. 90-102.
14. Sellin, R.H.J., Ervine, D.A., Willetts, B.B., (1992), "Behaviour of Meandering Two-Stage Channels", Proc. Of the Inst. Civil Eng., Water, Maritime and Energy, Vol. 101, No.2, pp.
15. Shiono, K. ,Muto, Y., (1998), "Complex Flow Mechanisms in Compound Meandering Channels with Overbank Flows", J. Fluid Mech., Vol. 376, pp. 221-261.
16. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1996), "Numerical Model Studies for Fish Diversion at Wanapum/Priest Rapids Development – Part I: Three-Dimensional Numerical model for Turbulent Flows through Wanapum Dam Tailrace Reach", IIHR

Limited Distribution Report No. 250, Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa, USA.

17. Stansby, P.K., Zhou, J.G., (1998), "Shallow-Water Non-Hydrostatic Pressure: 2D Vertical Plane Problems, Int. J. Numer. Meth. Fluids, Vol. 28, pp.541-563.
18. Speziale, C.G., Sarkar, S., Gatski, T.B., (1991), "Modelling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach", J. Fluid Mech., Vol. 227, pp. 245-272.
19. Tominaga, A., Nezu, I., Ezaki, K., Nakagawa, H., (1989), "Three-Dimensional Turbulent Structure in Straight Open-Channel Flows", J. Hydr. Res., Vol. 27, No. 1, pp. 149-173.
20. Willetts, B.B., Hardwick, R.I., (1993), "Stage Dependency for Overbank Flow in Meandering Channels", Proc. ICE Water Maritime and Energy, Vol. 101, No.1, pp. 45-54.
21. Younis, B.A., (2000), *personal communication.*

# Chapter 6:

# Application of CFD to Flooded Rivers

## – Rivers Severn and Ribble

# Chapter 6:

# Application of CFD to Flooded Rivers

## – Rivers Severn and Ribble

*"It is the engineer who must always be the link between the idea and actuality, between the probable and the practical."*

(Dumas, S. H., quoted in Lenox R. Lohr Centennial of Engineering 1852-1952)

### 6.1 RATIONALE

The previous section has demonstrated the capacity of Computational Fluid Dynamics (CFD) to reproduce detailed two-stage channel flow situations with a high degree of accuracy for the case of an idealised channel geometry. The physical soundness of the modelled processes has therefore been established, and it is now clear that such a numerical technique has the potential to simulate flows in natural channels. As already pointed out, three-dimensional modelling of river flows could be used to investigate velocity fields at structures, sediment and pollution transport as well as other environmental issues such as river restoration projects. In the particular case of river flood flows it would help bridge a gap into understanding the mechanisms controlling features of the velocity field in flooded meandering channels at large scale. This is important because the observation of such physical processes in real conditions is extremely difficult. It is apparent that, as flood flows are highly three-dimensional, such configuration should constitute a good test of the potential of CFD (complementary to that of Basara and Cokljat, 1995, and Wright and Morvan, 2000, for inbank flows) regarding river flow modelling in general.

In this chapter models of a reach of the Rivers Severn and Ribble are described. They attempt to reproduce recorded events measured during the course of this research work by the University of Lancaster, on some occasions with the assistance of the author. Because of the difficulty in collecting useful information on site, the density and quality of the data are lower than that from the Flood Channel Facility (FCF). However, provided a careful numerical verification is conducted, these could still give a first indication of the quality of the model's prediction.

The modelled processes will also be examined in the light of the FCF results, to assess their physical meaning, and determine whether any similarity exists between small- and large-scale processes. If obvious similarities were to appear, it could be concluded that the observations made at small scale in the FCF experiments are equally valid at a larger scale, and that the river model results are likely to be correct. On the other hand if the model seemed to match the field data but produced a significantly different flow representation from that in the FCF work, one would have to question the validity of the FCF conclusions. This second scenario would imply that more experimental work would be required to investigate the impact of model scale on the flow structure, prior to being able to conduct the validation of the CFD model.

## 6.2 RIVER SITES AND DATA

Three rivers were originally identified for this work because they present interesting plan view configurations, are easily accessible and, most importantly, are regularly flooded every year. One site located on the Nith nearby Dumfries in Scotland was later abandoned, as it proved too dangerous to undertake fieldwork at high flows. Most effort was therefore reported onto the other two locations described in more detail below.

### 6.2.1 River Severn Configuration

The River Severn is the third longest river in Britain after the Thames and the Wye respectively. It measures an estimated 206 km in length, drains an area of 4,330 $km^2$ and has a mean annual discharge of about 62.70 $m^3/s$ (Ward, 1981). It runs along the southeast border between England and Wales, and the Severn estuary is a significant landmark on the map of Southern Britain.

The section that is of interest to the current work is located 20 km east of Shrewsbury, Fig. 6.1. A single meander of about 600 m long, located south of Llandrinio, near Lower Farm was chosen for the project. The reach is shown in more detail on Fig. 6.2. At this location the main channel is about 30 m wide, between 6.0 and 7.0 m deep with respect to the upstream flood plain but more than 9.0 m with respect to the downstream left flood plain that has a higher elevation. The upstream right flood plain is 180 m wide and 120 m long, and is bunded by an earth embankment to the south.

The upstream right flood plain has been artificially lowered to extract material for the construction of the embankments, and is, as a result, fairly flat, Fig. 6.2. The upstream flood plain is easily flooded, which is why this site was chosen to obtain flood flow data. On the other hand the downstream left flood plain is rarely flooded, and it appeared during the course of this research that a very extreme flood would be required for it to be under water. This was unfortunate as it meant that the FCF channel layout – one meander with a "straight" overbank flow – could not be fully replicated at larger scale here.

### 6.2.2 River Severn Data

Over the past three years this reach has experienced a series of flood events. In particular events of approximately 100 $m^3/s$ have enabled the measurement of: water surface elevation along the reach; velocities at cross-sections and discrete locations; and turbulence data at discrete locations. In order to analyse the reach seven cross-sections were chosen along the course of the main channel. They are shown on Fig. 6.2 together with the location of the measuring tower, where detailed velocity and turbulence profiles are recorded, and that of the free surface measurements.

Five similar events of around 100 $m^3/s$ approximately were recorded in December 1999 (102 $m^3/s$), March 2000 (103 $m^3/s$), October 2000 (95 $m^3/s$), November 2000 (101 $m^3/s$) and February 2001 (104 $m^3/s$). In the case of the River Severn the discharges were estimated by integration of the measured velocities across the monitored section(s) by the field experimentalists at Lancaster University. This integration procedure was corrected to make sure it accounted for the total flow across the section, including both main channel and flood plain. Such procedure naturally entails a significant level of uncertainty in the determination of the exact discharges. However, the author cross-referenced these values with estimates of the water surface measured for each event at the monitored section(s) across the domain to ensure a reasonable consistency between the different events. The author checked that the water surface elevations were all consistent with the field data but also with TELEMAC free surface calculation.

Other events were recorded with sparsely distributed points, including the velocity field at one section for: a 85 $m^3/s$ flow in December 1999, a low 90 $m^3/s$ flow in October 2000 (inbank) and a 128 $m^3/s$ flow in December 2000. These were not used in the following work because they did not provide sufficient information to build a model in three dimensions. Further details concerning the field data and procedures should be provided in forthcoming report and CD-ROM by Beven, Carling and Holland from Lancaster and Southampton Universities.

The five 100 $m^3/s$ data sets mentioned above are merged in the steady state analysis conducted hereafter because of their similarity, and because they complement each other in terms of locations across the reach. Indeed sections in the centre of the meander, at the second bend and in the straight outlet reach were all measured for approximately 100 $m^3/s$, giving an overall picture of the velocity field in the domain. Although it is acknowledged that some discrepancy exists between the discharges and free surface profiles between each data set making them distinct events in effect, three main reasons justify the aggregation of the different sets:

(i) The 100 $m^3/s$ of December 1999 is the only event during which the free surface profile was surveyed across most of the reach, an information that can be used to construct and calibrate the models, but also to cross-reference measurements taken at discrete locations and at other dates ;

(ii) There is little flow data available to evaluate the models and each set alone is not sufficient to construct an entire model and obtain a clear picture of what is happening;

(iii) Having demonstrated that CFD models can be relatively accurate for open channel flows in Chapter 5, what is essential here is to ensure that the flow magnitudes and main trends are well calculated within the overall reach and for a given mass flow, at large scale in natural channels.

Most of the above five flood events are particularly interesting for a comparison with the outputs of Chapter 5 because they correspond to a depth ratio of about 20% to 25% on the right flood plain with the main channel depth. During the event of December 1999 (102 $m^3/s$), detailed free surface measurements were taken along the course of the channel and velocity field data were collected at cross-section 7. In October and November 2000 more data were collected in the region of sections 4 and 5 for similar discharges (95 $m^3/s$ and 101 $m^3/s$) and close water surface elevations. In March 2000 additional velocity and turbulence measurements were taken at the tower to provide data for the evaluation of momentum exchange at the interface between channel and flood plain.

During the December 1999 event the free surface was monitored along the northward and southward embankments from the upstream until the downstream end of the right flood plain, Figs. 6.2 and 6.3. The water surface build-up along the outer bend, at the first bend downstream of the inlet, is clearly visible on the collected data, Fig. 6.3(a). These data show that on the upstream flood plain the water surface in the west-to-east direction is practically flat but presents a mild slope to the south on the north-south axis. At the second bend, where the water converges in the "bottleneck", the flow is accelerated and a constant water surface slope of about $4.5 \times 10^{-4}$ is measured along the downstream straight, Fig. 6.3(b). This gives a clear picture of the free surface position across most of the domain. The velocity data of December 1999 shows a maximum velocity of 0.95 m/s in the lower half of the channel, Fig. 6.4, which is similar to data from March 2000 at the same location, Fig. 6.5. The autumn 2000 data, collected for similar water surface elevations at cross-sections 4 and 5, show the role of the flood plain water as a drive for the inbank flow, Figs. 6.6 to 6.8. For example the following data collected at section 5 in October 2000 show evidence of recirculation in the main channel (high negative angles in the lower right part of the channel, faster flow on the left hand side) driven by the flood plain flow:

| *Elev. (m)* | **Vel.** | 0.00 | 1.98 | 3.98 | 5.98 | 7.98 | 11.98 | 15.98 | 27.98 | 29.98 | 30.03 | 32.03 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| *18.26* | | 0.29 | 0.34 | 0.45 | 0.70 | 0.81 | 0.77 | 0.64 | 0.20 | 0.10 | 0.02 | 0.02 |
| *17.26* | | ■ | 0.34 | 0.45 | 0.70 | 0.81 | 0.77 | 0.64 | 0.20 | 0.10 | 0.02 | 0.02 |
| *16.26* | | ■ | 0.27 | 0.54 | 0.76 | 0.77 | 0.75 | 0.67 | 0.30 | 0.10 | 0.01 | ■ |
| *15.26* | | ■ | ■ | 0.58 | 0.73 | 0.76 | 0.76 | 0.67 | 0.36 | 0.11 | ■ | ■ |
| *14.26* | | ■ | ■ | 0.46 | 0.72 | 0.76 | 0.78 | 0.70 | 0.19 | 0.02 | ■ | ■ |
| *13.26* | | ■ | ■ | ■ | 0.76 | 0.75 | 0.77 | 0.60 | 0.21 | ■ | ■ | ■ |
| *12.26* | | ■ | ■ | ■ | 0.36 | 0.61 | 0.81 | 0.52 | ■ | ■ | ■ | ■ |
| *11.26* | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |
| *10.26* | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |

**TABLE 6.1 – Velocity Data (m/s) Collected at Cross-Section 5 on 29/10/00**

| *Elev. (m)* | **Ang.** | 0.00 | 1.98 | 3.98 | 5.98 | 7.98 | 11.98 | 15.98 | 27.98 | 29.98 | 30.03 | 32.03 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| *18.26* | | -12.1 | -9.8 | -4.3 | 1.9 | 15.1 | 14.2 | 6.7 | 10.6 | 8.1 | 45.0 | 37.5 |
| *17.26* | | ■ | -9.8 | -4.3 | 1.9 | 15.1 | 14.2 | 6.7 | 10.5 | 8.1 | 45.0 | 37.5 |
| *16.26* | | ■ | 6.1 | -3.9 | -1.6 | 2.1 | 2.9 | -3.1 | -2.3 | 3.4 | 45.0 | ■ |
| *15.26* | | ■ | ■ | -1.4 | -6.3 | -0.3 | -7.6 | -8.6 | -9.5 | 1.0 | ■ | ■ |
| *14.26* | | ■ | ■ | 0.1 | -7.1 | -2.8 | -16.4 | -17.5 | -5.8 | 5.2 | ■ | ■ |
| *13.26* | | ■ | ■ | ■ | -6.1 | -5.3 | -13.4 | -9.6 | -31.0 | ■ | ■ | ■ |
| *12.26* | | ■ | ■ | ■ | 8.6 | 5.3 | -9.5 | 7.9 | ■ | ■ | ■ | ■ |
| *11.26* | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |
| *10.26* | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |

**TABLE 6.2 – Flow Direction Data (deg.) Collected at Cross-Section 5 on 29/10/00**

Additional data were also collected at a measurement tower located immediately downstream of cross-section 7, on the right flood plain, at the interface between main channel and flood plain. Velocity and turbulence data were collected to analyse the turbulence momentum exchange between channel and flood plain flows. They are plotted on Figs. 6.9 and 6.10.

The data presented above will be used for the construction and validation of the numerical models. They will be used in a qualitative fashion to determine how well reproduced the physical features are, but also how the velocity profiles compare quantitatively at selected locations. They will later be available for further research on a CD-ROM compiled by the University of Lancaster.

### 6.2.3 River Ribble Configuration

The River Ribble is a much smaller river system than the Severn. It is 94-km long, drains an area of 1,140 $km^2$, has an annual mean flow of 31.72 $m^3/s$ and ranks $20^{th}$ in terms of length in Britain (Ward, 1981)

The site of interest for this work is located westward of Long Preston (on the A65 from Skipton). It forms a single Ω-shaped reach sandwiched between two embankments that nearly reduce inlet and outlet parts of the reach to straight inbank sections, Fig. 6.11. The main channel is about 1-km long, 25 to 30 m wide, 6.5 m deep, and the central flood plain is about 170 m wide by 110 m long, Fig. 6.12. The central flood plain is relatively flat and gets regularly flooded during autumn and winter seasons. A slight protrusion of earth runs along the inner bank around the second bend, separating the main channel and flood plain flows at low overbank depths.

From a numerical perspective this configuration could be interesting to test the advection and diffusion algorithms and turbulence models in the codes, in that large vorticity effects could be generated on the sides of the flow which "jets" from the inlet straight into a wide opening (flood plain) before exiting through a straight, narrow outlet. In addition the presence of the flood plain as a vertical obstacle in the centre of the domain makes it a true three-dimensional flow problem.

### 6.2.4 River Ribble Data

At this stage of the project few overbank flow data have been collected for the Ribble unfortunately, although some flood events were recorded by automated devices during the 1998-1999 winter. These records from the Environmental Agency (EA) have clearly identified several events of 80 to 100 $m^3/s$. In particular the 98 $m^3/s$ peak flood, shown in Fig. 6.13, was matched with a wrack line visible on the embankments and located "*half way up the bund*" according to the site experimentalist. In the absence of any other data this information was used in conjunction with free-surface simulations in two dimensions from TELEMAC to determine the position of the free surface for the entire domain.

## 6.3 NUMERICAL ISSUES ON MODELLING NATURAL RIVER CHANNELS

This section is going to review general issues regarding the application of Computational Fluid Dynamics (CFD) techniques to large, complex, natural river reaches in three dimensions. Although most of these issues were treated in Chapter 4, it was felt necessary to re-iterate some of these principles with regard to their application to natural river environments. Some of the strict physical and mathematical criteria behind CFD might need to be slightly compromised to enable the implementation of river models.

The geometry and scale of natural river channels require to be acknowledged as limiting factors regarding the application of the exact CFD principles given in Chapter 4. This is particularly true for the grid construction, the implementation of the law of the wall and convergence. One needs to determine the impact of compromising these principles on the model solution. This will be reviewed in sections 6.3.1 together with the presentation of a complete mesh independence test carried out with TELEMAC and CFX for the models of the River Severn.

Numerical discretization issues will also be addressed, and numerical simulation using first and third order accurate schemes will be compared to determine their impact on the solution.

A major problem when dealing with river modelling in three dimensions is the shortage of data, in particular at the boundaries. One therefore needs to discuss the implementation and usage of a considerable amount of complex information that data collection cannot provide. The generation of boundary conditions in this work will be briefly reviewed in section 6.3.3.

The criteria for convergence although similar to those presented in Chapter 4 will be further discussed in section 6.3.4 because it is likely that the intricate nature of the numerical grids will impair the efficiency of the numerical solver.

Finally a reminder of the turbulence models used by the author will be given in section 6.3.5.

### 6.3.1 Geometry and Mesh Resolution

6.3.1.1 Modelling the Geometry (CFX)

There are two essential issues that need to be addressed regarding modelling natural channel geometries in three dimensions (especially for CFX):

(i) The mathematical description of the domain geometry itself;

(ii) The mesh resolution that can be achieved and its numerical accuracy.

The accuracy of the domain representation can only be as good as the topographical data sampling permits. This issue implies that the numerical domain is only a simplified representation of reality, for which an approximate solution will be calculated. Since the geometry has to be constructed using discrete points, it is often difficult to build a "nice" geometry using sparse information, especially with a structured grid (CFX). The more irregular the geometry, the more difficult it also is to mesh. In complex domains the grid properties will consequently be seriously impaired by the nature of the domain. Poor grid properties, such as element poor aspect ratio, large growth ratio or lack of orthogonality of the grid lines with the boundaries, will in turn become a source of concern regarding the numerical stability of the model as well as its running cost. More iterations might be required for convergence. Turbulence models such as the Reynolds stress models (RSM) will be of limited use and the calculation of pressure at the boundaries will be inaccurate (Shyy and Vu, 1991) because the domain complexity compromises their stability. It might therefore be judicious to smooth the topography *a priori* and integrate some of the reach irregularities as extra roughness elements, at a latter stage, in the boundary conditions. Similar guidelines are provided in Sinha *et al.* (1996) and Shih *et al.* (1991). Unfortunately since little research related to river channels has been published to date, it is difficult to provide more accurate recommendations. The above difficulties are present in the models presented hereafter, e.g. in the Severn where the main channel shape varies significantly within the reach and has a narrow V-section in several locations which restricts the lateral distribution of nodes. However, the geometry of the problem cannot be changed! The user can only ensure that the shapes of the numerical domain are as smooth and simple as they possibly can, so that principles (i), (ii) and (iii), given in conclusion of section 4.3.1.2, can be respected. Such source of inconvenience will be dramatically reduced with the introduction of non-structured meshing to CFD codes, e.g. in CFX5.

The second issue (ii) can be addressed once the geometry is chosen, although the distribution of grid line and the spacing between grid points interacts with the properties of the geometry. The grid vertical and lateral resolution at the walls in particular, although smooth and progressive, will not be as fine in the model of a natural channel as they were in the CFX model of the FCF for example (see Chapter 5). They will be limited by the number of elements or nodes that can be included in the model (which is dependent on memory available on the computer). In the FCF, our numerical station experienced difficulties beyond 250,000 elements with CFX, and the grid CFX FCF-4, containing nearly 900,000 elements, could not be used at Glasgow University. Because of the added difficulty arising from the nature of the geometry, the author settled for a mesh size of up to 200,000 elements for the rivers modelled hereafter with CFX. In broad terms this represents about 20,000 elements for every 100 m of channel modelled. A simple test is therefore conducted to determine the impact of such grid resolution on the solution, using a simple trapezoidal channel with channel properties similar to the Ribble and Severn. More details of this test are given in section 6.3.1.2.

For TELEMAC the issues of geometry and grids are less problematic since the base grid is unstructured and the problem formulation can be seen as "two-dimensional". The description of the topography will be enhanced by structure lines and local refinement to ensure that as much detail of the topography as possible is included in the models.

6.3.1.2 Grid Resolution Test for Large Scale Models (CFX)

A simple test was devised to test the issue of mesh resolution in CFX, and in particular the impact of the vertical mesh resolution on the numerical solution in the centre of an idealised channel of large dimensions. Such tests should in fact be valid to evaluate the impact of the grid at all walls.

A trapezoidal channel of top width 30.0 m, bottom width 18.0 m and depth 6.0 m is constructed. The model is 100 m long and a resolution of 2.0 m is chosen in the longitudinal direction. The following cross-sectional resolutions are adopted:

| | **Mesh 1** | **Mesh 2** | **Mesh 3** | **Mesh 4** | **Mesh 5** | **Mesh 5** |
|---|---|---|---|---|---|---|
| ***Lateral Resolution*** | $r = 5.0$ ($r$ is the size ratio between the smaller and larger elements along one edge) $n = 20$ ($n$ is the number of elements) | | | | | |
| ***Vertical Resolution*** | $r = 1.5$ $n = 10$ | $r = 3.0$ $n = 10$ | $r = 5.0$ $n = 15$ | $r = 5.0$ $n = 25$ | $r = 10$ $n = 25$ | $r = 20$ $n = 50$ |
| ***Number of Elements*** | 10,000 | 10,000 | 15,000 | 25,000 | 50,000 | 100,000 |

**TABLE 6.3 – Grid Resolution for Large-Scale Model Test**

The first node is located at 78 cm, 55 cm, 26 cm, 15 cm, 9.5 cm and 2.5 cm from the bottom wall for meshes 1 to 6 respectively. The growth coefficient between consecutive cells along one edge is such that it is smaller than or equal to 1.2. This is what is recommended in codes such as CFX (Chapter 4). For simplicity a medium value is chosen for the lateral resolution as:

(i) The main flow gradient in the channel centre is over the vertical;

(ii) The observations made for the varied vertical resolutions at the bottom wall should be applicable to any wall.

In this model a uniform roughness value of $k_s$ = 0.100 m is chosen. It is comparable to what will be used in the applications shown in section 6.4. However with such a value, mesh 6 hardly manages to meet the requirements that $y^+$ < 500 (Versteeg and Malalasekera, 1995, p. 208). The mass flow in the channel is also comparable to what would be flowing in the channel for a depth of 5.0 to 6.0 m in the Severn and Ribble.

The results for the velocity in the channel centreline are shown in Fig. 6.14. All profiles are identical away from the wall; however, closer to the walls discrepancies are observed in particular with meshes 1 and 2. For these there is a clear break in the velocity profile, due to a poor application of the law of the wall. The following table illustrates the problem at the walls:

| | **Difference in Velocity Between Meshes:** (in % of Predicted Velocity with Mesh 6) | | | | |
|---|---|---|---|---|---|
| ***Distance from the Wall (m)*** | **1 and 6** | **2 and 6** | **3 and 6** | **4 and 6** | **5 and 6** |
| ***1.00*** | 4.24% | 3.54% | 2.88% | 1.76% | 0.66% |
| ***0.75*** | 6.33% | 4.06% | 3.48% | 2.20% | 0.85% |
| ***0.50*** | 6.65% | 6.20% | 4.01% | 3.16% | 1.24% |
| ***0.25*** | 43.48% | 23.29% | 5.63% | 6.02% | 2.28% |
| ***0.10*** | 70.71% | 63.53% | 48.76% | 31.65% | 5.60% |

**TABLE 6.4 – Comparison of Velocity at the Walls with Grid Resolution in the Large-Scale Model Test**

Such a sharp reduction in velocity at the walls would seriously compromise the simulation of sediment entrainment from the bed. The poor representation of the wall effects with Mesh 1 in particular is shown on Fig. 6.15. The bed shear stress is larger with this mesh than with Mesh 6 in the centre line (10% difference). It is also clear that the trough and peak discontinuity in the bottom corner is poorly captured despite a reasonably fine lateral resolution. The latter generates a difference of approximately 16% in the prediction of the shear stress. These results are all relative, since the flow is only parallel to the walls here and the channel geometry is relatively simple. In more complex situations, or with different wall roughness, these differences could be more significant.

It is encouraging however, that the velocity profiles away from the walls seem little affected. This means that a resolution such as that of Mesh 2 or 3 will probably be adequate in the forthcoming river applications, provided one accepts a large error in velocity within 25 cm of the walls and in the bed shear stress due to mesh coarseness. The difference between meshes 1 and 6 in the vicinity of the walls (about 40%) is fairly similar to that observed in the FCF between grids CFX FCF-1 and FCF-3, Fig. 5.34. The fact that the bed shear stress appears to be relatively independent of the level of discretization means that its prediction could at least be used in a qualitative manner to identify spatial variations of the stresses on the bed. Although uncertain, if the mesh resolution only resulted in a 10-15% error in prediction, it would still prove an accurate answer put in the context of what is customary in river engineering, particularly in the field of sediment transport.

6.3.1.3 Spatial Discretization (1: TELEMAC)

The details of the numerical meshes constructed in TELEMAC for models of the River Severn and River Ribble are presented hereafter. The quality of the different meshes, as well as their impact on the numerical solution, was investigated for both models. However, numerical details are given only for the model of the River Severn to avoid redundancy.

Grid Construction

Three numerical grids are constructed for each model in order to check mesh dependence.

First an unstructured surface mesh of 4,904 elements is built to discretize the reach of the River Severn, Fig. 6.16(a). A constant element size criteria of 10 m ($h$ = 10 m) is used to generate a regular mesh on the flood plain, while a criterion taking the slope into account is chosen on the channel banks to reduce element size in this location (a typical value is $h$

= 3.0 to 4.0 m longitudinally). Constraint lines are also used to incorporate the sudden change in the geometry profile in the mesh, notably at the banks and embankment, and restrict the application of the small element-size criteria to these regions of strong gradient. The level of detail embedded in the second half of the reach shows that the banks are particularly steep. The total number of nodes in the plan view is 2,796. Ten layers of the plan view grid are replicated over the water column to yield a three-dimensional grid made of 49,040 elements and 27,960 nodes. This mesh is called TELEMAC S-1.

Further refinement of the above mesh is carried out to assess the mesh independence of the solution. A second grid, TELEMAC S-2, is constructed aiming to improve the resolution of the riverbanks and main channel in particular. In this grid the typical element segment size is below 3.0 m on the banks and in the main channel. It results in a three-dimensional grid made of 41,200 nodes and 72,675 elements using a 10-layer discretization over the vertical, Fig. 6.16(b). This grid is as fine as what other TELEMAC users involved in detailed CFD work have used recently (Hankin *et al.*, 2001), yet finer than what most practitioners would be willing to implement.

Finally, for comparison further refinement is added to the grid to try and increase the numerical resolution at the side walls again and reach that of CFX grids for the same reach. The new grid, TELEMAC S-3, consequently carries 66,830 nodes and 117,261 elements spread over 10 vertical layers. Its typical grid spacing at the banks is below 2.0 m. This grid is simply used for testing purposes here.

Similar work was conducted for the TELEMAC model of the Ribble, which results are presented in section 6.5. These results were used to estimate the scaling curve (CPU time vs. number of nodes) presented in Figs. 4.22 and 4.23.

<u>Mesh Independence</u>

Cross-sectional analysis of the mass flow is conducted for the three grids previously described for the model of the Severn to ensure that mass-conservation is globally met in the domain. Sections taken across the upstream part of the reach where the right flood plain is flooded give more approximate answers because the mesh is coarser on the flood plain. Yet for TELEMAC S-1, the coarser grid, the error in discharge is only of 2.5%, which means that a much coarser grid could have been used while ensuring a reasonable accuracy. The error in mass is reduced to about 1% with the finer grids across the upper half of the reach. At the second bend, a similar calculation indicates that TELEMAC S-1 and TELEMAC S-2 show an error of 1.5% in discharge, whereas TELEMAC S-3 is accurate to within 1%. Further downstream, towards the outlet section, the accuracy is within 1%, probably because the main channel conveys most of the flow. The error observed on the flood plain is due to a numerical approximation during the integration. Where the grid gets finer, in particular in the neighbourhood of the main channel, the integration is very accurate. Such accuracy shows that all three grids provide more than an adequate level of mass-conservation. It is certain that a much coarser grid would have been sufficient to obtain a general picture of the flow in two dimensions for example.

The different grids were compared using the two-dimensional solutions from TELEMAC-3D since the code is fundamentally providing a solution for two-dimensional horizontal layers. All three simulations were conducted with the same distribution of roughness values discussed in section 6.4.1.4. It should be noted that contrarily to what was pointed out by Hankin *et al.* (2001) the enhancement of mesh resolution did not appear to affect the choice of roughness calibration values significantly here. At the scale of the reach, it is likely that the downstream boundary condition – obtained from field observation – dominates the determination of the free-surface position and apparently restrains its sensitivity to roughness and grid size.

A comparison of the spatial distribution of discharge therefore is conducted to see how the flow fields calculated on the three grids compare. About one hundred of the point data analysed along the main channel thalweg, the right flood plain and across sections 4, 5, 6 and 7 are plotted on Fig. 6.17 for illustration purposes. Most of the difference in mass flow at any given location between the grids is well within 10%, with the largest discrepancy located at the banks and in region of shallow waters. Larger random differences are visible on Fig. 6.17, however this is limited to a few data points and the correlation coefficient is equal to 99% for both plots. Further statistical analysis was conducted on the full data set this time and indicated that the 95% confidence level from the mean was of 97.78% and 97.38% between the outputs of TELEMAC S-1 with S-2 and TELEMAC S-1 with S-3. This means that there is more than 97% chance that the answer calculated with any two of these grids will be the same within an error margin of ±5%.

6.3.1.4 Spatial Discretization (2: CFX)

As in the previous section, most of the details regarding how the numerical grids were constructed for CFX are given for the example of the River Severn. It goes without saying that what was undertaken for one river model was also done for the other.

Grid Construction

Two numerical grids with different node densities are constructed for each river model after a suitable structure of blocks has been implemented to describe the channel morphology, Fig. 6.18. However, the number of elements is deliberately kept slightly lower than for the FCF, as the complexity of the geometry will generate additional difficulties that are likely to require a higher processing power. In addition, small spurious elements might also be created, as the grid becomes finer. They are known to be a source of difficulty, especially in the numerical treatment of turbulence terms at the walls, which could cancel out the benefits of the finer resolution by impeding convergence.

The process of refining the grid to assess the level of mesh dependence assumes that one knows in advance where grid enhancement would be most needed, and therefore which are the problematic flow features likely to be poorly resolved. If one did not know these features, one would have to identify them to optimise the choice or refinement approach. In the present case however, it is anticipated that one of the problematic features is going to be the sharp gradients of velocities, at the walls and where flows cross, as identified in the FCF work.

The first grid constructed for the Severn is made of 97,732 elements. It has 10 elements positioned across the channel and 14 over the vertical, both with a growth ratio of 1.5. There are only 4 elements over the vertical in the flood plain area because the water depth is about 1.5 m. The mean resolution in the domain is therefore of 0.6 m on the vertical and 1.8 m laterally and 2.0 m in the channel longitudinal direction. The first element is typically located at a distance of 0.77 m from the wall in the vertical direction and 0.80 m in the lateral direction at the bottom.

Further refinement is conducted to create a second grid made of 183,138 elements. This is done by the enhancement of the lateral and vertical distributions so that 17 nodes are created on the vertical and 15 laterally in the main channel, and 5 elements vertically on the flood plain. This represents an averaged resolution of 0.5 m in the vertical direction, 1.2 m in the lateral direction and about 1.6 in the longitudinal direction. The resolution at the bed walls is now of 0.40 m on the vertical and lateral directions, which represents a considerable improvement on the first grid but is still not sufficient to obtain a low $y^+$ value close to 500. This grid has similar properties to meshes 2 and 3 in the example shown in section 6.3.2. These two grids are called CFX S-1 and CFX S-2 respectively and are detailed in Table 6.5.

Mesh enhancement is difficult to conduct in the case of the River Severn because any cross-sectional refinement entails a costly refinement in the longitudinal direction due to the block arrangement, Fig. 6.18. This difficulty stems from the reach layout, in which the main channel forms an "S" shape that contains the flood plain, and therefore cannot be described by a simple structure of blocks running parallel to the channel. Instead the multiblock description of the channels leads to "nested" elements on the flood plain. A CAD design tool my have allowed more flexibility in the design of the geometry and enabled a better grid to be constructed.

In addition to the above difficulty, the cross-sectional shape of the Severn also provides an additional complication. Its V-shape means that it is difficult to obtain a fairly regular resolution over the channel width with the depth, especially in places close to the bed where the section is particularly narrow and where bottom and side walls intersect, Fig. 6.19. Indeed whenever a fine resolution is chosen across the channel width spurious elements are created at the bed where side and bottom walls intersect, which prevents the use of the mesh. This phenomenon is aggravated by the channel curvature. This explains why the lateral resolution is kept so coarse despite the main channel top width of 30.0 m. Resolutions such as that of meshes 3 to 6, as presented in section 6.3.2, could not be implemented here. Instead it was attempted to reproduce a sound grid with characteristics similar to that of mesh 2, so that a reasonable approximation to the velocity field could be obtained throughout the domains (at the exception of the walls) at a "low" computational cost.

| | **Model of the SEVERN** | |
|---|---|---|
| ***Location – Direction*** | **Mesh CFX S-1** | **Mesh CFX S-2** |
| ***Main Flood plain – Lateral***<br>*(on right side)* | 95 × 50 elements<br>$r$ = 1.0; *width × length* | 120 × 100 elements<br>$r$ = 1.0; *width × length* |
| ***Flood plain – Vertical***<br>*(above bank level)* | 4 elements<br>$r$ = 1.0 | 5 elements<br>$r$ = 1.0 |
| ***Main Channel – Lateral*** | 10 elements<br>$r$ = 1.5 (*) | 15 elements<br>$r$ = 3.0 (*) |
| ***Main Channel – Vertical***<br>*(inside main channel only)* | 10 elements<br>$r$ = 1.5 (*) | 12 elements<br>$r$ = 3.0 (*) |
| ***Main Channel – Longitudinal***<br>*(with effects on all domain)* | 337 elements<br>$r$ = 1.0<br><u>*along main channel*</u> | 425 elements<br>$r$ = 1.0<br><u>*along main channel*</u> |
| ***Total*** | 97,732 | 183,138 |

(*) *indicates a bi-directional bias, see chapter 4;* (BL) *from the bank level in the upward direction,* (MC) *from the main channel towards the flume sides.*

**TABLE 6.5 – CFX Grid Characteristics Used to Model the Severn Reach.**

In the case of the River Ribble, the simpler trapezoidal channel layout is a main advantage and allows a much better lateral and vertical resolution. For example the resolution in the main channel is 13×20 (depth × width), which implies that the mesh structure is likely to enable a better implementation of the boundary conditions at the side walls. However the combination of tight bends and shallow depth with gently sloping banks in part of the channel slightly impairs the construction of the grids. It appears that the construction of small elements along the bend inner walls results in the collapse of some elements, due to the intersection of the grid line with the wall geometry. This is particularly the case where the bank geometry presents a convex bulging shape directed towards the inner side of the domain, as in the first bend in the Ribble reach. This problem is illustrated on Fig. 6.20, together with the aspect ratio concern at the intersection between side and bottom walls, when the angle becomes larger than 30° (see recommendations in Chapter 4).

Mesh Independence

As for the FCF model, the non-dimensional pressure coefficient, $C_p$, is compared for each set of grids along the centreline of the main channel. In the case of the River Severn these simulations are carried out for a flow of 100 $m^3/s$ (Fig. 6.21). The solutions can be described as being very close. In fact it seems that the numerical scheme has more impact of the pressure solution (see 6.3.2.2) than the grid resolution. In terms of the velocity field, comparisons between the outputs of the two grids at the seven monitored cross-sections show little difference. As expected, the velocity is better resolved at the walls with the finer grid, and the difference in velocity between CFX S-1 and S-2 at the walls is similar to that between meshes 1 and 2 in section 6.3.1.2. The difference in velocity intensities elsewhere is negligible, and the difference in direction is also limited to a few degrees as was observed in the FCF (Figs. 5.34 and 5.35).

This means that at the exception of the walls the solution of grid CFX S-2 is relatively mesh-independent. A completely accurate solution cannot be obtained throughout the entire domain due to issues of scale, high roughness and complexity of the geometry which would demand a grid resolution at the walls that is unpractical. At the walls the velocity is therefore likely to be unpredicted, as shown in section 6.3.1.2. As in section 6.3.1.2, the bed shear stress seems little affected however, Fig. 6.59. Identical tests were conducted for the model of the River Ribble, and yielded similar comments although more emphasis was put on the resolution of the main channel in that model.

The grids employed here are coarse compared to what is usually the case in CFD, due to the size and complexity of the domain. The concept of "mesh-independence" is therefore relative. As pointed out by Lane *et al.* (1999), it can even be described as "*nebulous*" in natural channels, especially as the finer the grid the more dependent the solution becomes to the spatial sampling used to collect the topographical data. This point was illustrated earlier by the author, in the form of equation (4.14).

### 6.3.2 Numerical Discretization

6.3.2.1 TELEMAC

The operator-splitting method described in Chapter 4 is implemented. The optimum discretization set up is implicit in TELEMAC to exploit the Navier-Stokes equation properties in a finite element framework and maintain mass-conservation.

Consequently a finite element discretization is applied to the calculation of the water depth (SUPG for the advection part and standard Galerkin method to the diffusion part). The Method of Characteristics (MOC) is applied to the advection part in the momentum equation for the calculation of the convection velocity, followed by a Galerkin formulation for the diffusion part of the momentum equation.

6.3.2.2 CFX

CFX offers more choices, especially to discretize the advection part of the Navier-Stokes equations. Simulations were therefore conducted using both the CCCT and hybrid schemes for the advection terms, in order to determine their impact on the solution. The results presented hereafter are for the model of the River Severn, for a mass flow of 100 $m^3/s$.

A comparison between the results for the two numerical schemes at cross-sections 3 to 5 is shown in Fig. 6.22. Some difference exists between the two, yet it is not that significant. CCCT predicts a slightly faster flow in the meander, but the differences remain small and localised (about 5% of the maximum cross-sectional velocity), which makes it difficult to visualise in the figures. Similar findings were made with the model of the River Ribble where maximum differences of 7% (equivalent to 5cm/s) where found in the cross-over region (sections 5 and 6), Fig. 6.12. From a river engineering perspective such level of difference in the solution is minor.

It is generally recommended to use high order scheme when investigating detailed fluid flows to alleviate false diffusion (Easom, 2000). However, for the level of spatial resolution and the accuracy employed in this work the author did not notice excessive numerical diffusion. Similarly, other researchers successfully implemented upwind and hybrid schemes in their work, e.g. Sinha *et al.* (1998) for a model of a 4-km reach on the Columbia River, or Wu *et al.* (2000) to model sediment transport in a scale model. This comforted the author in his conclusion that the current river models do not justify the adoption of a high order numerical scheme to investigate the cross-sectional velocities. This is quite important for practitioners, as hybrid is, in general, not only less expensive to use, but also more robust. It is especially important with the multi-block approach used in this study, since it also simplifies the calculations conducted at the block interfaces and reduces the level of interdependence between the blocks.

### 6.3.3 Boundary Conditions

#### 6.3.3.1 TELEMAC

As presented in Chapter 4, the inlet boundary condition in TELEMAC is implemented using the subroutine `Q3D` suggested by the manual. This subroutine underlines the two-dimensional background of the code as a uniform depth-averaged velocity is applied at the inlet. Such boundary condition has been repeatedly implemented in varied river and estuarine flow situations by TELEMAC users and developers (Peltier *et al.*, 1996; Bates *et al.*, 1998). It constitutes an adequate condition for the algorithm since the first step of each iteration consists of a two-dimensional solution of the flow in order to determine the water depth necessary to the calculation of the mesh elevation. It is also a condition that can be easily implemented by most practitioners since it does not require detailed information about the flow. Additionally a logarithmic velocity profile is found to develop quite rapidly over the vertical after the inlet section leading the author to assume that the boundary condition is satisfactory. One could chose to set the inlet velocity condition node per node, however this would be laborious and, in the absence of detailed inlet velocities, it is not worth the trouble. Furthermore the manual indicates that such set up would require access to the source code to be really efficient (Janin *et al.*, 1997b). Sensitivity to TELEMAC boundary conditions has been recently documented by Lancaster researchers (Hankin *et al.*, 2001), however, since the dominant factor of the parameterisation problem at reach scale with TELEMAC seems to be the friction term (Bates *et al.*, 1998) it received priority treatment. No testing of the inlet boundary conditions is conducted in the present work, and the discharge provided from the field is used to implement `Q3D`.

At the downstream boundary the water level is implemented. This can be done using `Q3DSORTIE` (Chapter 4) for an unsteady flow or simply by setting the water level at a constant value. In the case of the steady 100-$m^3$/s flow computed hereafter for the River Severn, it is set to approximately 18.40 m from the field data. For the River Ribble, it is set to 10.25 m based on field evidence as well. The outlet discharge is calculated by `Q3DSORTIE` to monitor mass conservation through the domain and ensure that steady state is reached.

The boundary conditions at the walls and at the free surface are as presented in Chapter 4. Roughness is implemented in TELEMAC-3D in the form of a Chézy C value calculated from an approximate knowledge of the bed roughness or estimated from site observations by the author. In the following river application the set-up of spatially varying roughness values has to be managed by the user via a FORTRAN subroutine in the boundary condition file. The subroutine written by the author attributes a roughness value to each node on the bottom layer as a function of their spatial location using the table `RUGOF(NPOIN2)` (Janin *et al.*, 1997a). In order to obtain a more realistic, yet manageable, distribution of roughness values across the reaches a simple assumption is made. Since the reaches modelled here are short and most measurements were taken in autumn and winter (short vegetation), one roughness value is assumed for the flood plain and another for the main channel.

6.3.3.2 CFX

A simple logarithmic profile based on the depth of water is implemented at the inlet in CFX. Its parameters are adjusted to yield the required discharge and obtain a realistic velocity profile compared with field data or information available from TELEMAC simulation (Wright and Morvan, 2000):

$$U_{in} = C_{in} \ln (z/z_0) \quad \text{with } z_0 = {}^{k_s}\!/_{30} \tag{6.1}$$

For the Severn $C_{in}$ (equivalent to $u_*/\kappa$ ) is taken equal to 0.080 m/s, which corresponds to an average shear velocity of about 3.2 cm/s. In the main channel this yields velocities of up to 0.65 m/s at the inlet. This is close to observed values collected by Lancaster University and found with TELEMAC. Similarly a constant value equal to 0.100 m/s is calculated to fit the mass flow for the Ribble. It is easier in the latter case and of less consequence upon the overall solution since the inlet is narrow. This constant is equivalent to a shear velocity of 4.1 cm/s and equation (6.1) then generates a maximum velocity of 0.75 m/s close to the surface. For comparison, simulations using a mean velocity across the inlet boundary were attempted but systematically failed, due to instability at the bottom and side walls stemming from the incompatibility between the high velocity and the high roughness at the walls. The type of inlet profile provided by equation (6.1) is consequently necessary for numerical reasons as well.

Ideally one should have implemented a more detailed velocity field via the FORTRAN subroutines. Unfortunately such detailed data was not available to the author. Therefore an artificial way of implementing a velocity profile had to be devised. It should be noted that this is likely to be the case in the vast majority of river engineering projects. It is difficult to say how accurate the profile generated using equation (6.1) is with respect to the river sites studied here, however it has a general sound physical basis and the reaches are long enough to allow the flow to develop before reaching the meanders. Furthermore detailed work carried out in Chapter 5 (section 5.4.7.1) regarding the sensitivity of the solution to the inlet boundary conditions has indicated that rigid lid models are relatively insensitive to the boundary conditions. Similarly Alfrink and van Rijn (1983) have also suggested that variations of the inlet flow and turbulence conditions had a limited impact on the solution of a flow over a trench. These remarks suggest that equation (6.1) should provide a sufficient boundary condition, and its impact on the solution is therefore not investigated further.

Concerning the implementation of the turbulence inlet conditions the reader is referred to equations (4.18), (4.20), (4.22) and (4.24) in Chapter 4. Assuming a shear velocity of about 3.2 cm/s for example, equation (4.20) would yield an average turbulence kinetic energy value of $1.707\times10^{-3}$ $m^2/s^2$ while (4.18) gives $0.867\times10^{-3}$ $m^2/s^2$. Equations (4.20), (4.24) and (4.22) are applied without further investigation, based on the conclusions of Chapters 4 and 5, for the turbulence kinetic energy and the energy dissipation at the inlet, and the calculation of the dissipation length scale.

At the outlet a mass flow boundary condition is implemented, as in equations (4.14) and (4.15). This assumes fully developed flow conditions.

At the walls an algebraic law similar to (4.35) is implemented. In itself the law has been shown to be perfectly suited for high turbulence flows, such as in natural flooded rivers, in Chapter 4. Because of the coarseness of the grid though it is expected that the first node might be located too far from the wall, and that the merging of the law of the wall with fully turbulent flow condition be inadequate in the region of the walls (Fig. 4.9). Wall roughness is implemented with separate roughness height values for the flood plain and main channel for the same reasons as with TELEMAC.

The free surface is modelled as a rigid lid, which is either constructed from knowledge of the water surface elevation data collected on site or from simulations conducted with TELEMAC. The rigid lid is in fact an impermeable shear-free wall. Pressure is monitored on the lid and its position adjusted manually in an iterative fashion using this information. The task of adjusting the lid over such complex domain is quite difficult and localised errors might be impossible to eliminate fully. Such adjustment is conducted in parallel with the calibration process of setting the wall roughness value, since roughness determines the pressure gradient (equivalent to free surface gradient in open channels) on the lid. This will discussed case per case in sections 6.4.2.2 and 6.5.2.2.

### 6.3.4 Convergence

#### 6.3.4.1 TELEMAC

Since steady-state flows are modelled in this work, simulations in TELEMAC can be stopped once the outlet discharge yields a constant value equal to the inlet discharge and when the variables at the nodes are calculated to within 1 cm/s using the two-dimensional solver (depth averaged nodal value). A similar procedure was implemented in the FCF work, and was also recently described by Horritt (2000) in a paper based on a river flood application with TELEMAC.

#### 6.3.4.2 CFX

The convergence criteria formulated in section 4.2.5 for CFX are applied. It should be said, however, that a reduction of the residuals by four orders of magnitude is difficult to achieve here, despite the use of the Algebraic Multigrid (AMG) solver (see sections 4.3.5.3 and 4.3.5.4). Consequently, some simulations were stopped between $10^{-3}$ and $10^{-4}$, which is still compliant with Mesehle and Sotiropoulos (2000); in doing so the AMG outperformed the other methods, which either failed or did not converge to that level within the same time. This underlines the interest in using such a reliable type of solver with complex domain.

The author observed that convergence was more difficult when extremely smooth or rough boundary conditions were used at the walls. As repeated by Abbott (1989), "*instability is the number's way of telling us that our scheme contains contradictory statements*". In the present context it was inferred that instability resulted from physically incompatible boundary and flow conditions. The converse implies that a well-converged solution is the sign of a well-formulated problem with consistent boundary conditions.

In some cases turbulence models also appeared to be a source of instability. As indicated in Chapter 2, RSM is known for its stiffness and convergence difficulties, which were aggravated by the geometry to make its use impractical for the reach of the River Severn in particular.

### 6.3.5 Turbulence Models

Chapter 5 has illustrated the fact that in geometry-dominated flows such as that of the FCF, turbulence appeared to be secondary in the determination of the flow structure. Basara and Younis (1995) have also reported that in the case of the calculation of the flow over a trench, the accuracy of simple turbulence models was adequate for engineering applications. This is an important issue here since it is likely that most of the numerical effort should be dedicated to the calculation of a sound numerical solution in a complex domain. There is also an issue of resolution involved in the decision to use complex models: Is it worth employing a sophisticated and costly model where the discretization might be inadequate and the level of uncertainty in the description of the problem is larger than turbulence-induced effects?

In the following work, the author will consequently focus on the application of the mixing-length (TELEMAC) and $k$-$\varepsilon$ (CFX) models, which were found to be adequate in the case of the FCF model. Results obtained with the RSM for the River Ribble will be presented in Section 6.5.

## 6.4 RIVER SEVERN

Two models of the Severn reach presented in section 6.2.1 are constructed hereafter. First, a TELEMAC model is constructed to assess the position of the free surface and the impact of calibration on the solution in comparison with measured data. Part of this information is then used to complement that obtained from the field data to build a fully three-dimensional model of the reach with CFX. The solutions provided with both codes are analysed, validated and finally compared in section 6.4.3.

### 6.4.1 Quasi-3D Model using TELEMAC

6.4.1.1 Determination of the Wall Roughness

The Ramette formula (4.55) recommended by TELEMAC developers is used to relate the roughness element size at the walls to the value of Chézy C (Janin *et al.*, 1997). In the case of the studied reach, the formula indicates roughness values between 30 and 50 $m^{1/2}/s$, calculated from knowledge of the bed material composition measured as varying between 4 and 14 mm in the main channel. These calculations are detailed hereafter.

In the main channel, grain material at the bed is reported to vary between 4 and 14 mm. Assuming that the roughness height is directly proportional to the grain size, the formulae presented in section 4.3.3.4 would yield $k_s$ values of between 30 and 100 mm. For the main channel, the application of Massey's formula (4.38) suggests an equivalent Mannning's $n$ of about 0.026, and therefore a Chézy C of about 45 to 50 $m^{1/2}/s$ ($C = R^{1/6}/n = 5.5^{1/6}/0.026 \approx 50$). Typical Manning's $n$ values for flat, pasture flood plain have been reported to be about 0.030 (Chow, 1959). A reverse calculation would therefore suggest values of the order of 35 $m^{1/2}/s$ to 40 $m^{1/2}/s$ on the flood plain, where the flow depth is approximately 1.5 m during the 100 $m^3/s$ flood event. ($C = R^{1/6}/n = 1.5^{1/6}/0.030 = 36$).

This range of values is consequently used in section 6.4.1.2 where the calibration process is reported.

6.4.1.2 Sensitivity Analysis and Calibration

The main focus of the following tests is to determine a suitable set of roughness coefficients for the observed flow and establish how sensitive the solution is to changes in this boundary condition.

The roughness parameter impacts on the slope and position of the free surface, and on the velocity field in TELEMAC. In a first stage a uniform roughness value is applied to the entire domain. This is because a preliminary analysis has shown that main channel and flood plain roughness values seem relatively close, and because little field data is available on the flood plain to determine the true position of the free surface. Collected data indicate that the water surface elevation is constant and equal to about 18.49 m in the upper part of the channel, along the left bund. In the lower part of the channel the data show that the water surface profile presents a constant slope of about $5.9\times10^{-4}$, starting at about 18.48-18.49 m from the previously described flat upstream profile.

Fig. 6.23 offers a detailed comparison between the water surface elevations measured in December 1999 along the upstream section (left channel bank) and along the downstream end (along the right bund) for only a few of the roughness values tested by the author. It shows that the downstream change in the free surface slope is reasonably well captured by the models, as are the two slopes. However, it is obvious from the graph that $C = 45$ m$^{1/2}$/s is too smooth, and that $C = 35$ m$^{1/2}$/s seems the most adapted to that section, although the largest discrepancy between the two is probably only about 3.5 cm. In the upstream section, it is clear that the hydrostatic pressure assumption built into TELEMAC does not allow the capture of the pressure build up around the first bend. All profiles are very flat until after the bend where they incline slightly. The best fit for the section would seem to be $C = 45$ m$^{1/2}$/s to 40 m$^{1/2}$/s. A comparison of the calculated velocity outputs shows that the rougher the flood plain, the slower the flow on the flood plain and the faster the flow in the main channel.

Using this information it is attempted to apply a more realistic distribution of roughness, assuming the flood plain roughness to be higher than that in the channel calculated in section 6.4.1.1. Different combinations of values are used, assuming a flood plain roughness of 35 m$^{1/2}$/s to 40 m$^{1/2}$/s and a channel roughness equal to 45-40 m$^{1/2}$/s. The results are shown on Fig. 6.24. The combination "$C = 45$ m$^{1/2}$/s in the channel and $C = 40$ m$^{1/2}$/s" on the flood plain is apparently too smooth and yields an upstream water level (<18.48 m) which is too low compared to the field observations. On the other hand a slightly rougher flood plain ($C = 35$ m$^{1/2}$/s) combined with a channel roughness of 45 m$^{1/2}$/s yields a water surface profile that seems quite adequate, equal to 18.49 m in the upstream part. Such a different distribution of roughness also impacts on the velocity distribution in the domain.

The combination "C = 45 $m^{1/2}/s$ in the main channel – C = 35 $m^{1/2}/s$ on the flood plain" yields a relatively slower velocity on the flood plain but a faster velocity in the main channel compared to any of the other presented cases, Fig. 6.25. The velocity is 11% slower than the faster velocity on the flood plain and 10% faster than the slower velocity in the main channel, which represents an average difference of 3 cm/s on the flood plain and 5 cm/s in the main channel. Practising engineers should be aware that such differences exist in the distribution of velocities between roughness set-ups when large domains are modelled. In particular, although the plots for C = 40 $m^{1/2}/s$ and "C = 45 $m^{1/2}/s$ in the channel – C = 35 $m^{1/2}/s$ on the flood plain" yield a very similar water surface profile, their velocity distribution is very different. Indeed they exhibit differences of 4.0 cm/s on the flood plain and 3.2 cm/s in the main channel. Fig. 6.25 shows that as roughness increases on the flood plain a faster velocity occurs in the main channel while it is slower on the flood plain. If the roughness of the main channel is then slightly increased, this trend is reduced and a new balance between the flow velocities in the main channel and the flood plain is reached. The model could therefore be calibrated for the velocity field and the water surface profile assuming sufficient information was available for both. Unfortunately the lack of data does not permit such calibration here. In the author's opinion it would be also the case in practise, where most models are usually adjusted for the free surface profile. This is a heritage of one-dimensional modelling in industry, however, the previous analysis has illustrated that distributed models would require a more complete calibration.

In the following analysis, the set up "C = 45 $m^{1/2}/s$ in the main channel – C = 35 $m^{1/2}/s$ on the flood plain" is used because it allows a good match with the field data, and seems in accordance with the evidence collected on site regarding the nature of the wall boundaries.

### 6.4.1.3 Predicted Velocity Field and Flow Mechanisms

In the next paragraph, the author attempts to analyse the model's outputs from a qualitative point of view, with reference to the Flood Channel Facility (FCF) research programme. A numerical comparison against field data will be carried out in the next section to validate the model's predictions.

Fig. 6.26 shows a depth-averaged vectorial representation of the flow calculated with TELEMAC two-dimensional algorithm. The main channel flow can be seen cutting the edge of the right bank at the first bend, which greatly influences the velocity pattern in the central part of the upstream flood plain. Along the left bank, the flow is travelling more slowly and running parallel to the wall. At the second bend, water depth is quite shallow on the right flood plain, and there is a contraction of the flow. Further downstream, the flow is much faster (larger vectors) as a result of the bottleneck effect and narrowness of the channel past the second bend.

The analysis is continued by looking at the flow pattern in more detail at various cross-sections, along the course of the channel. The location of these sections is shown on a layout of the river reach, Fig. 6.2.

Cross-section 1 displays the computed velocity profile 80 m downstream of the inlet, Fig. 6.27. It is mostly used to check the inlet boundary condition and adjust the model with field information available for the velocity at the inlet.

Cross-section 2 repeats a similar view further downstream, before the first bend. The velocity in the main channel is close to 0.50 m/s in the main channel. It is also faster along the right bank and on the right flood plain. This is where the inbank flow cuts the corner at the first bend and starts interacting with the flood plain flow. A strong recirculation starts to form as shown in Fig. 6.31; it reaches an intensity equivalent to about 15-20% of the local maximum velocity.

Past the first bend, cross-section 3, the model shows that the velocity maximum is shifted to the left, where it starts to dive downwards, Fig.6.27. This is evidenced on the angle plot, Fig. 6.29, which suggests that the flow rotates to the right as over the height of the water column. This phenomenon is of course accentuated by the flood plain flow coming from the right, which helps shift the velocity maximum against the left bank, and initiate rotation within the walls of the main channel. The main channel geometry forces the inbank water, and part of the flood plain flow, to rotate in a counter-clockwise fashion, Fig. 6.31.

Cross-section 4, Fig.6.27, could be reminiscent of the flow pattern at cross-section 8 in the FCF, although there is no opportunity for the flow to escape on the downstream flood plain here. Undoubtedly however the flow is vigorously stirred and rotated at this location, as both the angle and velocity (e.g. on the right bank) plots show, Figs. 6.27 and 6.29. Secondary currents are calculated to reach an intensity of 25% of the normal cross-sectional velocity maximum computed to be about 0.50 m/s, Fig. 6.31. Rotation is also implicitly shown in the upper part of the right bank in the shape of the circular isovel.

The results at cross-section 5 show an illustration of a flow similar to that at cross-section 8 in the FCF, Fig.6.28. The flood plain flow reaches the main channel at a 50-60° angle, Fig. 6.30, which results in a strong rotational inbank flow entrained by the shear generated by the overbank flow. The shape of the 0.60-m/s isovel is indeed identical to the 0.30-m/s isovel in the FCF, Fig. 5.46. The intensity of recirculation stretches as far as 30% of the maximum velocity, Fig. 6.32, a magnitude found and exceeded in the FCF. However, it seems that the flood plain flow coming from the right at sections 4 and 5 is comparatively slower than the incoming flow in the FCF: It represents only about 40 to 70% of the main channel velocity in the model of the Severn whereas it was 100% in the FCF.

At cross section 6, Fig.6.28, the flow exits the second meander and the peak starts to shift slightly towards the channel centre. There is still some obvious evidence of anti-clockwise rotation in the vertical plane, however it is much weaker (intensity of 12.5%), Fig. 6.32. The flow is now mostly travelling in the longitudinal direction, as the low angles and quasi-symmetric angle plot show, Fig. 6.30.

Further downstream, at section 7, Fig.6.30, there is little rotation within the cross-section. The flow is re-aligned with the main channel, at least in the upper part of the channel. The isovel plot, Fig. 6.28, is inaccurate, as it shows no sign of turbulent momentum exchange at the interface between the main channel and the flood plain (see Figs. 6.4 and 6.5 for a comparison). An anisotropic turbulence model would have been required to model non-equal normal Reynolds stresses at this interface.

6.4.1.4 Model Validation against Velocity Data

This section aims to present a comparison between the model's prediction and some velocities collected in the field, in particular during the winter 2000-2001. This should validate the model's ability to represent the physics of the flow quantitatively. The focus of this comparison is on bend sections 4 and 5 for which collections of data were carried out for flows close 100 $m^3/s$ in November and December 2000, and in February 2001.

The comparison between the computed (Figs. 6.27 and 6.28) and measured (Figs. 6.6 to 6.8) isovels is reasonably good in the region of cross-section 4 and 5. At section 5, the model predicts a velocity maximum in the region of 0.80 m/s at 10 m from the left bank. Towards the right bank the isovels split between overbank and inbank regions. In the main channel the velocity is around 0.55 m/s on the right hand side whereas it falls to 0.40 m/s closer to the bank level, as observed in the field. This indicates that TELEMAC captures the main flow features in the region of the cross-over, as it did in the FCF. However, when more detailed quantitative comparison is made between calculated and measured velocity profiles some discrepancies appear.

At cross-section 4, the velocity along the left bank is reasonably accurate, especially in the region close to the free surface, Fig.6.33. Deeper the acceleration in the longitudinal direction (due to the flow diving and turning) is modelled but greatly attenuated, and results in a 25% difference with respect to the data set dated 15 December 2000. In the channel centre, the range of velocity seems reasonable compared to data set from 13 February 2001, although the bulging shape of the velocity profile is hardly predicted close the bottom. This is also true, when the model is compared with the data from 15 December 2000. Such difference between the two field data sets illustrate how difficult it is to collect data in the field, and how dangerous it might be to merge the two sets for the sake of analysis.

Similar comparisons are made for the velocity profiles at cross-section 5, Fig. 6.34. The predicted velocity along the left bank of section 5 is satisfactory, and the slight bulging profile below the free surface is also accounted for. However, it decreases rapidly and leads to a 30% underestimate of the maximum velocity close to the bed, reminiscent of the errors found at the bed with coarse grids in section 6.3.1.2. This is due to a problem of resolution close to the bed in the model. Closer to the centre of the channel, the proximity between data and model is quite clear, although there is a marked difference between 14.0-m and 16.5-m depth (approximately 25%). Along the right bank, the predicted and measured profiles do not compare very well, despite yielding a very similar depth-averaged velocity. The model seems to under-predict the velocity in the upper part of the water column, while it over-predicts it further down. This could be due to a difference in the prediction of the flow direction, in the upper part of the channel in particular, which would affect the profiles.

At cross-section 7 the difference between predicted (Fig. 6.28) and measured velocity profiles (Figs. 6.4 and 6.5) is obvious. This section of the reach behaves like a straight two-stage channel, and the turbulence model employed here is responsible for most of the difference. The use of an anisotropic turbulence model should improve such calculation (see section 5.4.9).

The measured data have helped demonstrate that there is some reasonable correlation between observations and predictions, albeit with a large relative error in places, notably at the walls, which is normal. Part of this error can certainly be attributed to a degree of inaccuracy in the data, which is difficult to quantify, and to uncertainty in the model's construction. Other errors also result from the attempt to calculate large vertical velocities, thereby violating the hydrostatic pressure assumption built in TELEMAC. It seems however that TELEMAC has the potential to satisfactorily represent the flow main features in the cross-over region, even if it should be stressed that the above comparison is insufficient to constitute a proper validation.

### 6.4.2 Fully-3D Model using CFX

6.4.2.1 Determination of the Wall Roughness

The determination of bed roughness in natural channels is a difficult issue. In the case of the Severn, the particle diameters were found to vary between 4 and 14 mm. Although roughness has been related to grain roughness for historical reasons, there is little knowledge about the relationship between grain size and roughness height for such gravel bed material as underlined in section 4.3.3.4. Moreover Carling (2001) recently stressed that one should not systematically look at grain roughness for such applications, since there are many other parameters other than grain roughness that influence the overall roughness, e.g. bedforms or turbulence. However, in the absence of any better knowledge, the author considered the different relationships discussed in Chapter 4. These yielded values of $k_s$ between 5 mm (pure grain roughness; too smooth according to Wu *et al.*, 2000) and 30-100 mm (grain and small bedforms). Such information gives at best an indication of what roughness could be, based on the results of past experimental work, Clifford *et al.* (1992) for example.

Earlier calibration work carried out using the free surface code TELEMAC can also be used to back-estimate the equivalent roughness height. TELEMAC results have indicated that a suitable range of roughness values would be a Chézy $C$ = 45 $m^{1/2}/s$ in the main channel and $C$ = 35 $m^{1/2}/s$ on the flood plain. This set up roughly corresponds to a roughness height in the region of 0.100 m for both flood plain (shallow water) and main channel (coarse material), which fits with some of the estimates calculated above from the knowledge of the grain size.

The values that have been previously calculated are large, yet smaller than in Sinha *et al.* (1996) for example. They result from an application of grain size – roughness relationships outside their range of validity, and/or the use of formulations that include several momentum loss mechanisms that are not properly described in a lumped roughness coefficient. This underlines the need for more research in this area, from an empirical and numerical point of view. During the course of this work, values within the

0.100 m order of magnitude are used as an initial guess, yet their physical meaning is unclear. In the author's mind they are mainly coefficients used to define momentum and energy losses within the model at the walls.

6.4.2.2 Sensitivity Analysis and Calibration

Having established in Chapter 5 that the inlet/outlet boundary conditions have a limited impact on the solution in the case of a rigid lid model, simple boundary conditions are applied at the inlet, outlet and free-surface, as described in section 6.3.3. On the other hand a thorough investigation of the impact of the wall roughness on the solution is attempted for the event of December 1999, during which water surface profiles were collected. Since the channel is much rougher than in the case of the FCF it is also suspected (Chapter 5) that the roughness height values should have more impact on the velocity profile. The variation of the velocity profiles to change in roughness at the seven cross-sections is consequently monitored.

Roughness values are determined following the indication provided in section 6.4.2.1. In a first stage a unique $k_s$ value is applied in a uniform manner. For the purpose of the following analysis a simulation run with $k_s$ = 0.100 m is taken as reference. Field measurements taken along two lines on either side of the flood plain limit with the embankments, Fig. 6.2, are used for comparison with CFX pressure outputs, Fig. 6.35. Along the first meander the water surface elevation is well reproduced: The flat profile at 18.485 m is correctly calculated, and so is part of the water surface elevation build up at the bend. The difference at the downstream end is of the order of 1.5 cm.

A second simulation is run with $k_s$ = 0.005 m to simulate a very smooth wall boundary. As can be seen in Fig. 6.35 the downstream surface profile on the flood plain is very close to the data. On the other hand the difference is less marked along the upstream bend for different roughness values, which corresponds to the fact that the pressure profile calculated by CFX is in fact flatter here. Indeed when comparing the next cases with higher roughness values, the similarity in the profiles is obvious which reminds us that the roughness value adjusts the pressure slope (rather than the "water elevation") on the fixed lid. This is one of the model shortcomings which would suggest that such sensitivity analysis test is a mere tuning of the model boundary conditions (roughness, inlet mass flow, lid position) to fit the unique event that is modelled. With such a low roughness the velocity profile at cross-section 7 is quasi uniform, Fig. 6.37, as expected, and does not compare with reality at all. It should be reported that convergence is more difficult to obtain with low roughness values, and that subsequent adjustments in the reduction and relaxation parameters are required.

Another two simulation are run with $k_s$ = 0.200 and 0.300 m. With $k_s$ = 0.300 m it is clear that the water profile tilts downwards too much (Fig. 6.36) and this condition seems too rough. On the other hand as roughness increases beyond 0.100 m the velocity fields shows little evolution, and in particular the velocity maximum in the main channel remains desperately lower than the observation reported (0.77 m/s). With larger values than 0.200 to 0.300 m convergence becomes difficult to achieve. The author suggests that the different conditions applied in the model might become inconsistent, in particular the position of the rigid lid with the chosen roughness for the 100 $m^3$/s flow attempted here, and the model breaks down.

In general the model appears rather insensitive to changes in the roughness values within the large range of values tested above. The use of different roughness values for the flood plain and the main channel, within the calculated range, does not significantly influence the numerical solution in terms of velocity and pressure fields. However, large values are needed for the model to operate satisfactorily, which suggests that energy-dissipating mechanisms are not well described at the walls. The pressure profile obtained with $k_s$ = 0.100 m (Fig. 6.36) is in accordance with the TELEMAC free-surface outputs. Additionally, the range of pressure on the lid is lower here than the results obtained by Sinha *et al.* (1996, Fig. 3.1 p. 88; see Chapter 3), in which pressures of the order of 1,000, Pa were still found on the lid after calibration.

6.4.2.3 Predicted Velocity Field and Flow Mechanisms

A model of the river Severn has been built for the December 1999 flood event of 100 $m^3/s$, described and calibrated in the previous sections. A plan view of this flow close to the free surface is shown in Fig. 6.38, as calculated by CFX. The flow can be seen cutting across the first bend, turning and contracting immediately upstream of the second bend, and finally accelerating in the downstream part of the reach. These general features are similar to those calculated with TELEMAC earlier for the same flow. The velocity field is then analysed in more details at cross-sections 1 to 7.

Cross-sections 1 and 2, Fig. 6.39, show the velocity profile implemented at the entry for the main channel, with a maximum velocity of 0.55 m/s to 0.45 m/s close to the free surface. The flow is also quite fast on the flood plain where there is evidence that it is cutting the corner (see Fig. 6.38). This would explain the slight erosion of the inside meander visible in the topographical profile of the flood plain bed.

At cross-section 3 the flow is exiting the meander along the outside wall, because the bend is extremely tight here. This results in an early rotation of the flow from the top to the bottom in an anti-clockwise direction, as visible on the angle plot, Fig. 6.43. There is some additional evidence of this on the velocity contour, as illustrated by the stretched shape of the 0.40-m/s isovel along the right bank. At this stage the flood plain flow is running with a 20° angle with the flow in the main channel forced to turn by the geometry of the bend. On the flood plain the flow is approximately normal to the cross-section however. The recirculation is about 20% of the computed cross-sectional velocity maximum, Fig. 6.43.

Cross-section 4 looks similar to the cross-over section in the FCF. The flood plain flow intrusion now distorts the isovels along the right bank, and results in a curved velocity iso-contour line, Fig. 6.39. Strong lateral velocities distort the velocity profile to the right at the bed, and point to a strong rotational motion. This clearly indicates the generation of a helical motion of a similar nature to that of the FCF, although the higher left bank probably constrains it by diverting part of the flood plain flow in the direction of the main channel. This results in a lower top lateral shear from the flood plain flow onto the inbank flow (70% of inbank velocity), and the stretching and weakening of the helicoid in the main channel direction. Analysis of secondary motion in the channel centre indicates a recirculation cell of intensity equal to about 20% of the local maximum velocity, Fig. 6.43.

At cross-section 5, the velocity is visibly increased due to the sudden reduction in the channel area, Fig. 6.40. Because of its set up cross-section 5 really constitutes the best equivalent to the cross-over region in the FCF terminology: The flood plain flow is crossing over the main channel before continuing in a straight direction on the left flood plain downstream. Additionally, the overbank flow is the region of cross-section 5 is travelling at an angle of 70° with the inbank flow, Fig. 6.42, and the depth ratio is about 20 to 25%, nearly as in FCF experiment B23. As noted with the TELEMAC model the flood plain flow is slower than in the FCF compared with the inbank velocity, 40% only. However, the rotational motion is still very strong: The lateral velocity is close to 0.25 m/s just below the surface, which corresponds to an intensity of 30% of the main channel velocity, Fig. 6.44. The fast surface flow coming from the right flood plain rolls over the inbank flow. Some water is ejected on the left bank where it is accelerated to reach 0.75 m/s, and redirected to travel at an angle of 10° with respect to the normal to the cross-section. Such features are identical to earlier comments made by the author regarding the FCF flow mechanisms at cross-section 8 (Chapter 5). Assuming the current model to be correctly implemented, this would indicate clear similarities between the FCF and natural river flow features.

The velocity structure at section 6 is similar to that of section 5, however, there is evidence of a reduction of intensity in the anti-clockwise rotation because there is no or little lateral inflow, Fig. 6.44. This is confirmed in the angle and the intensity of secondary current figures, where it can be estimated that the secondary velocity in the main channel is now of the order of 18-20%, although the shear flow across the section close to the surface still seems strong. Because of the position of cross-section 6 with respect to the upstream shear flow the helicoid receives no lateral top shear and is stretched in the downstream direction, which further weakens its intensity.

Further downstream, at cross-section 7, secondary currents are very weak, of the order of 5-6% of the cross-sectional velocity maximum, Fig. 6.44. This indicates that the helical motion dies rapidly when the channel is re-aligned. The flow is travelling in one main direction at cross-section 7 and there is no marked distinction between channel and flood plain flows, Fig. 6.40, although field evidence and theory suggest it should be otherwise. This is characteristic of what was observed during the reproduction of Tominaga's experiment (Chapter 5), which is most probably due to the turbulence model inability to account for the momentum exchange across the interface between main channel and flood plain. This failure accounts for a lower maximum in the velocity field at this location (discussed later). In that respect this data indicates the need for an anisotropic model. It does not imply that CFD has failed in the present case, but that the turbulence closure model chosen here is insufficient to describe the physics of the flow.

It was therefore decided to implement the Reynolds Stress model (RSM) described in Chapter 2 to resolve the above problem. However it was found very difficult to get the solution to converge properly, and the attempt was aborted since the author had little confidence in the quality of the numerical solution.

6.4.2.4 Numerical Tracers

Numerical Tracer experiments were conducted to follow particles released from the upstream end of the reach, Figs. 6.45 to 6.48, and confirm the interpretation of the flow dynamics presented above. Particles released in the main channel travel mostly in the channel or its immediate proximity. There is evidence however that these particles are not only entrained along the channel but are travelling in a more complex fashion. Some of them can clearly be seen to cut the meander and to rotate. To do so they are transported out of and back into the main channel, Figs. 6.45 and 6.46. Particles located along the right bank, before the second bend, can be seen to be transported across the channel and towards the left bank through the second bend. These features confirm that the inbank flow is rotating on itself and that the drive for such rotation is the flood plain cross-flow. Particles released at mid-depth from the left hand side of the upstream flood plain seem to be plunging into the main channel in the contraction region (cross-section 6), whereas particles released along the right embankment appear to be travelling in a straight direction across the channel meander, Figs. 6.47 and 6.48.

Another interesting feature from these experiments is that a recirculation appeared to be reproduced right after the second meander due to the channel expansion to the left. Although they were unable to measure it, site experimentalists did report observing a horizontal recirculation in this region.

6.4.2.5 Model Validation against Velocity and Turbulence Data

The previous discussion has illustrated the existence of some similarities between the observations made during the FCF programme and the current model's predictions. The mechanisms driving the flow are close, however further quantitative validation is required to ensure that the velocity magnitudes are also correctly reproduced.

A comparison between the model, Fig. 6.40, and the data collected at cross-section 5 in November 2000, Figs. 6.6 and 6.7, indicates that model and data display similar velocity fields. A fast flow in the region of 0.75-0.80 m/s is the visible in the left upper half of the channel on both series of plots, while a velocity core of about 0.75 m/s is running along the left bank. In the data from November 2000, this core is slightly inclined to the left as predicted in the CFX model where it can be seen to "lean" against the bank slope due to the flood plain flow lateral shear. In the lower right part of the channel the measured velocities reach 0.50 m/s as predicted by the model, while they are reduced to 0.40 m/s, against 0.30 m/s for CFX, in the upper part. For the event of December 2000 recorded between cross-sections 4 and 5, Fig. 6.8, similar comments apply when considering the predicted flow at sections 4 and 5 in the model, Figs. 6.39 and 6.40. This suggests that the physics of the flow is well captured by the model around the meander.

A full cross-section was also monitored at section 7 during the 100 $m^3$/s flood flow event of December 1999. The comparison between data and prediction is not as good as in the meander above, yet it is most certainly stemming from a modelling error associated with the choice of turbulence model. Turbulence momentum exchange at the interface between flood plain and main channel is highly anisotropic and greatly affects the velocity profile in straight reaches. This is consequently a distinct error (a modelling error) from any of the differences previously noted for sections 4 and 5 for example. It results in the calculated velocity maximum to be lower than the measured value by about 15% in the main channel, and there is no marked distinction between channel and flood plain flows, as should be the case in a straight two-stage channel.

A more objective analysis compares some of the velocities measured over the water column depth in the region of cross-sections 4 and 5 in November and December 2000, and in February 2001 with the model's predictions, Figs. 6.49 and 6.50. From these figures it is clear that there exists larger discrepancies than previously expected between data and predictions at given locations. The order of the difference is about 20 to 30%. However, the shape of the velocity profiles is also reasonably reproduced at most locations and a significant part of the difference is at the walls. In general CFX under-predicts the velocity field at most locations.

Part of the difference between predictions and observations could be explained by the fact that one is attempting to compare a simulation for a steady flow with a 100-$m^3$/s discharge with different unsteady flood flows with approximate, fluctuating discharges. It could indeed be stemming from an erroneous prediction of the discharges from the field data, as the discharge values were determined by integration of the velocity measurements. An attempt is made to try and resolve part of this uncertainty by running the model for a discharge of 120 $m^3$/s. The results with CFX are better and the match between the velocity profiles visibly enhanced at cross-section 5 for example, Fig.6.51. However improving the velocity at section 5 results in a slightly faster flow at cross-section 4, and raises the question of the model's calibration for such a flow (notably the pressure on the lid). If the test clearly suggests that errors in the measured discharge could account for the difference between model and data, it does not imply that the model is faultless. At best it highlights how difficult it is to assess a three-dimensional model quantitatively with only restricted data such as the water surface profile, approximate discharge values and a few velocity measurements in such a complex geometry.

During the event of March 2000, detailed velocity and turbulence measurements were taken at the tower located on the right flood plain close to the outlet, Fig. 6.52 and 6.53. Unfortunately these data were collected where the author has already stressed that the model produces inadequate predictions, due to a modelling error of turbulence. What is clearly visible from the comparison of the model with the data is that the velocity predictions are less accurate in the region of the channel-flood plain interface, with errors of up to 30%. Further away from the interface the predictions are more in agreement with the data (within 15%). Such drift could be critical for sediment transport problems however. In terms of turbulence kinetic energy (TKE), the model presents a lower, somewhat uniform TKE across the direction of measurements, whereas the data indicates an increasing value of TKE from the main channel on the flood plain. Errors of 100% are visible at the bed at most locations, which rises the question of the accuracy of the bed shear stress carried out in section 6.3.1.2 and 6.4.2.6 hereafter. Large level of error (50%) and a similar pattern of under-prediction for the TKE have also been reported by Bradbrook *et al.* (1998), although for a model of a laboratory experiment.

Within the two research groups involved in this project discussions have been conducted regarding the meaning of the measured turbulence data. Two explanations have been put forward regarding the nature of the turbulence intensity, which the author reproduces below. These could explain the shape of the observed profiles and, in part, some of the differences between the CFX model and the data:

(i) The team lead by Prof. Carling believes there exists a two-layer flow at the interface between main channel and flood plain, as found in some of the FCF Series A experiments. In the upper part of the water column, lateral shear would dominate leading to a more uniform distribution of TKE, while vertical shear dominates at the bed resulting in a higher production of TKE that an isotropic turbulence model fails to capture.

(ii) Some data also suggested to the Glasgow University team that the flow is decelerating at the tower (due to a pool at the bed). Evidence presented by Graf (1998) suggests that turbulence intensity is very sensitive to changes in velocity, and that when the flow is decelerating turbulence is greatly enhanced. This deceleration effect might not be well captured in the CFX model, which would accentuate the difference between data and prediction.

From these discussions it has emerged that it is very difficult to use turbulence data from the field, in particular as uncertainty in the data, the friction factor and the frame of reference seem to dominate their interpretation. Additionally the scarcity of the data leaves them open to interpretation and speculation.

6.4.2.6 Predicted Bed Shear Stresses

Bed shear stresses were calculated at the seven chosen cross-sections, using the calculated turbulence kinetic energy data at the walls. They are plotted on Figs. 6.54 to 6.60.

At cross-sections 1 and 2, the bed shear stress is fairly uniform, around 0.4 N/m$^2$, with a slight increase up to around 1.0 N/m$^2$ in the main channel, Fig. 6.54 and 6.55. At section 3, in the first bend, the bed shear stress increases up to 2.0 N/m$^2$ in the angles of the cross-section, Fig. 6.56, and at section 4, the value on the left bank reaches 2.2 N/m$^2$. This is probably the result of the flow being redirected towards the contraction section 6 in the plan view, combined with the effects of the emerging helical motion. At Sections 5 and 6 the bed shear stresses reach 5.5 N/m$^2$ left bank due to the effects of a very strong anti-clockwise secondary cell in the main channel, Figs 6.58 and 6.59. At section 7, Fig. 6.60, the bed shear stress is lower on the left bank, however it is quite large on the right flood plain.

Bed samples taken on the bed at varied locations in the main channel of the Severn, reveal that a typical bed element has a diameter of 11mm ($D_{50}$) and 14 mm ($D_{84}$) in the second bend, while it is only 4 to 6 mm everywhere else in the reach, including in the first bend.

Using Shields' results (French, 1985), it is possible to estimate the approximate local critical shear stress:

$$\tau_{cr.} = 0.0924 \times \rho g d \approx \frac{\rho g D}{11} \qquad (6.2)$$

Where $\rho$ is the density of water, $g$ the Earth gravity field and $D$ the mean bed material diameter. For a typical diameter of 6 mm for the reach, for which equation (6.2) is valid, the critical shear stress is about 5.3 N/m$^2$. This means that particle entrainment should occur in the neighbourhood of sections 5 and 6 according to the model, which predicts a maximum shear stress of 5.5 N/m$^2$ at the foot of the bank. At sections 5 and 6 strong diving currents are induced with the potential to generate a large stress at the foot of the bank and the entrainment of 6.0-mm bed particles. This explanation would indeed be consistent with the fact that the bed material at this location is made of larger particles that have not been eroded (They would require a critical shear stress of 10 N/m$^2$). Assuming a quasi-linear relationship between bed shear stress and particle diameters one would expect from the sediment material found on site that the bed shear stress in the inlet, the first bend and the outlet should be 2.5 to 3.0 times smaller than at sections 5 and 6, exactly what is predicted by the model.

Since it was noted in Chapter 5 that CFX models seemed to capture well the calculation of the extremes at the walls, and following the results of section 6.3.1.2, the current simulation should display a sound physical process. The latter would explain scouring at and the collapse of the left bank at section 5 (Fig. 6.61). Calculations carried out with both grids (CFX S-1 and S-2) have shown little change in the bed shear stress, Fig. 6.59, as in section 6.3.1.2, despite a marked difference in their resolution at the walls. Furthermore, evidence presented by Lane *et al.* (1999) suggests that if the prediction of bed shear stress is highly sensitive to the choice of roughness value in two-dimensions, this is less the case in three-dimensions where the dominant effects of vertical motion upon bed shear stress are dealt with explicitly.

The above observations would suggest that the phenomena predicted by CFX are consistent with physical evidence, although its numerical accuracy remains more speculative. This implies that CFD could at least be used to examine complex bed shear stress distribution. Compared to what one- and two-dimensional models currently offers, the variations shown in Figs. 6.58 and 6.59 would make a significant difference in the prediction of sediment transport which behaviour is known to be highly non-linear (Bettess, 2001).

### 6.4.3 Models of the River Severn: Conclusions

The general picture obtained from the above simulations has shown some striking similarities with the results from the FCF experimental work concerning both meandering and straight compound channels. Indeed secondary currents at the meander in the model of the Severn exhibit magnitude and generating mechanisms similar to those observed in FCF experiment B23 (SERC, 1993). It seems that, like in the FCF, the flow structure is mostly geometry-driven and dominated by the flood plain flow depth, velocity and angle with respect to those in the main channel. Comparison of TELEMAC and CFX models against field data confirms that the flow mechanisms are correctly accounted for where inbank and flood plain flows interact in the region of sections 4 and 5 for the Severn. The general features of the velocity profiles (velocity magnitude and approximate location of the isovels) seem to be well reproduced, although both models use very simple turbulence models.

These results are encouraging since they would suggest that:

(i) River flood flows in meandering channels behave in a similar fashion to their scale model counterparts. Laboratory results, such as that of the FCF, could therefore be used to calculate flood flow dynamics. Their only drawback would be the fact that they were obtained for smooth walls, whereas this study has illustrated the potential difference associated with coarse, complex surfaces.

(ii) As in the FCF, turbulence would appear to play a secondary role in the determination of the three-dimensional flow field in the meanders of flooded rivers. This is not true in straight sections, as seen for section 7, however.

(iii) CFD could be used to investigate detailed velocity profiles in the field of river engineering, away from the walls;

(iv) CFD could also be used to identify irregular bed shear stress pattern in spite of restrictions imposed by the scale of the problem on the meshability of the domain.

Both TELEMAC and CFX make very similar predictions (at the exception of section 3), which is comforting, although CFX is fully three-dimensional and accounts for non-hydrostatic pressure effects. Although not very significant in appearance for the prediction of the velocity field here, the calculation of pressure is necessary to obtain detailed velocity and turbulence profiles, notably at the walls in the cross-over region or over topographical irregularities in general (Stansby and Zhou, 1998), and therefore for sediment transport problems (Fukuoka and Wanatabe, 2000). Stansby and Zhou (1998) further demonstrate that the steeper the bank profile (1:2 vs. 1:5), the more important is the need for a full calculation of pressure. Obviously, the calculation of pressure is necessary where strong vertical velocities invalidate the hydrostatic pressure assumption. This does not reduce the value and usefulness of TELEMAC to adequately represent the main features of the flow at a low cost. The code shows its in-built ability to deal with natural, large-scale river flows, whereas CFX suffers from a more rigid approach, in particular in its ability to deal with the problem geometry.

Quantitatively however, both models have shown that one could expect an error of 20% to 30% in the prediction of velocities at one given location, and that in most cases the velocity field was under-predicted. This has been attributed, in part, to the inaccuracy in the field data and in particular to an inaccurate calculation of the mass flow from the sparse field data. It is clear that more data, and a better knowledge of the overall mass flow, would have helped narrow the source of uncertainty in the predictions. The models are certainly not without faults either, and it has to be said that the CFX predictions are not better than TELEMAC's. In particular, in the region of the second meander, the CFX model clearly under-predicts the velocity compared the field data. Part of this difference could be due to an inaccurate positioning of the lid and bed profiles, offering a larger cross-sectional area in places, and therefore an underestimate of the velocity. Because the data were not collected in a structured fashion, the construction of a structured geometry made of 6-face "cubical" blocks could have indeed resulted in additional errors in the domain representation for CFX. Yet, such uncertainties make it impossible to fully assess the performance of the models, as was possible in the FCF.

Evidence collected on the site seems to confirm the validity of the bed shear stress prediction, at least its order of magnitude (CFX). To be able to reproduce such features numerically and at large scale would imply that one could investigate bank erosion processes using CFD techniques incorporating all flow components in time and space. On the other hand the prediction of the turbulence kinetic energy (TKE) at the tower was not very good. Although this could be due to the choice of turbulence model and to the sensitive nature of such measurements in the field, it could also indicate that the TKE and bed shear stresses at the bed are miscalculated, at least in some places.

Regarding models' CPU performance, TELEMAC unstructured grid and simpler algorithm enable the model to outperform CFX. The model run using TELEMAC S-1 only required 25 hours to converge, whereas up to 65 hours where necessary with CFX S-1 to reduce the residual by 3 to 4 orders of magnitude on a Sun station Ultra-10/433 with 384 MB. Despite the multiblock approach and the use of a multigrid algorithm, CFX4 has difficulties to deal with the nature of the channel morphology because of the structured grid. It is expected that in the case of more regular geometry configurations, such as trapezoidal channels, CFX4 algorithm would perform better. Additionally it would be interesting to test the new unstructured version of CFX (CFX5) in a similar situation.

## 6.5 RIVER RIBBLE

Two models built with TELEMAC and CFX are presented in the next sections, following the same layout as for the Severn. However, in the absence of field data, the interest of this section is mostly to present an additional study case to illustrate the application of CFD techniques to river flows, and draw some qualitative comparisons with the FCF.

TELEMAC is used to provide the free-surface position necessary for the construction of the CFX model here. Therefore more emphasis is put on gaining a better understanding of the flow mechanics in the Ribble, rather than on the comparison between the benefits and flaws of the two models and codes.

### 6.5.1 Quasi-3D Model of the Ribble using TELEMAC

#### 6.5.1.1 Determination of the Wall Roughness

In the Ribble reach, the central flood plain is relatively flat and covered with short pasture grass. As a consequence, a value of Manning's $n$ = 0.030 seems appropriate to represent the flood plain roughness (Chow, 1959). It corresponds to roughness height values of 0.200 m to 0.100 m in the case of a 98 $m^3/s$ flood event, because the water depth is relatively low on the flood plain.

Investigation on site has revealed that the main channel bed is covered with a sandy material, which would make the bed roughness relatively smooth, $0.020 \leq n \leq 0.030$ probably. According to Ramette's formula used in TELEMAC (Janin *et al.*, 1997), this would correspond to low roughness height values of the order 0.030 to 0.080 m, for the flood flow of 98 $m^3/s$ modelled hereafter. In comparison with the values used in the model of the Severn, this seems reasonably consistent. Indeed, assuming a linear relationship between grain size and roughness height, such roughness height would imply bed particles of millimetric size (1.0-2.0 mm) which correspond to sands.

The roughness values previously discussed correspond to Chézy C of 35 $m^{1/2}/s$ to 40 $m^{1/2}/s$ on the flood plain and 50 $m^{1/2}/s$ to 55 $m^{1/2}/s$ in the main channel.

#### 6.5.1.2 Sensitivity Analysis and Calibration

Data collection in the River Ribble has been very poor, and only one out-of-bank flow was reported in an approximate fashion. Calling the following "calibration" is therefore rather ambitious, since the choice of the model parameters is largely the result of the author's own interpretation of the site and flow features.

Observations from October 1998 reported a flow of approximately 98 $m^3/s$ leaving a wrack line half way up the bund. An analysis of the topographical data carried out *a posteriori*, showed that the bund is only about 0.80 m to 1.00 m high, and sloping very gently towards the downstream end. Combining both pieces of information lead the author to reconstitute the position of the wrack line in the frame of the survey data. At the downstream end, it corresponds to an elevation of 10.25 m for example. In the region of the central flood plain "*half way up the bund*" is equivalent to an elevation of 10.55 to 10.60 m approximately.

Early simulations using uniform Chézy $C$ values of 40 $m^{1/2}/s$ to 50 $m^{1/2}/s$ for the entire domain seem to indicate that this model is more sensitive to variations in the boundary conditions than the model of the Severn, Fig. 6.62. The author observed variations of water level of up to 17.0 cm, between extreme cases, at the upstream end of the reach. Such difference is important in the case of the Ribble because it corresponds to 20% to 25% of the flood protection embankment height.

However, once one settles for the range of values established in Section 6.5.1.1, assuming the main channel to be relatively smooth, the variability of the free surface position can be reduced. In particular the simulations using $50 \leq C \leq 55$ in the channel and $35 \leq C \leq 45$ on the flood plain all meet the objective of an elevation equal to 10.60 m on the central flood plain. What is noticeable then, is that increasing the flood plain roughness mainly impacts on the position of the free surface on the flood plain, which becomes higher and flatter. This shows that with more detailed data it should be possible to adjust the position of the free surface in the main channel and on the flood plain quite independently.

As previously reported for the Severn, there are cases where the position of the free surface is quasi-identical despite the use of completely different set ups (e.g. "$C$ = 50 $m^{1/2}/s$ in the main channel – $C$ = 45 $m^{1/2}/s$ on the flood plain" vs. "$C$ = 55 $m^{1/2}/s$ in the main channel – $C$ = 38 $m^{1/2}/s$ on the flood plain"). The occurrence of equifinality cases for CFD multi-parameter models has only been discussed recently (Hankin *et al.*, 2001). Their difference lies in the distribution of the mass flow, i.e. the velocity. This input should consequently be used to calibrate the model. Fig 6.63 illustrates very well the impact roughness has on the velocity field. It is indeed clear that a general trend is for the velocity on the flood plain to decrease as roughness increases. Increasing the roughness on the flood plain and decreasing it in the main channel accentuates this trend. The difference in velocities is large, of the order of 15% to 20% on the flood plain, and about 11% in the main channel (excluding the case " $C = 50\ m^{1/2}/s$ in the main channel – $C = 35\ m^{1/2}/s$ on the flood plain"). Comparing the cases "$C = 55\ m^{1/2}/s$ in the main channel – $C = 38\ m^{1/2}/s$ on the flood plain" and "$C = 50\ m^{1/2}/s$ in the main channel – $C = 45\ m^{1/2}/s$ on the flood plain" of interest here, a difference of 12% (7.0-8.0 cm/s) can be calculated for the flood plain peak velocity, whereas it is about 8% (5.0 cm/s) in the main channel. Since there is such a difference in the velocity field, but that no velocity data is available to make a decision, the author feels that two set-ups could have been used.

#### 6.5.1.3 Predicted Velocity Field and Flow Mechanisms

The calibration process has illustrated the varying nature of the velocity field in the present model depending on the calibration set up. Results using the set up "$C = 55\ m^{1/2}/s$ in the main channel – $C = 38\ m^{1/2}/s$ on the flood plain" are presented next for a $98\ m^3/s$ flow.

A depth-averaged plot of the velocity field vectors calculated with TELEMAC for the Ribble is shown in Fig. 6.64. This figure clearly shows the flow being ejected and accelerated onto the central flood plain immediately downstream of the first bend. From the vector scale it is possible to see that the faster flow occurs along the west embankment. On the flood plain the streamlines follow the path imposed by the boundary lines of the west embankement. In the vicinity of section 6, the flood plain flow running along the west bund crosses over the main channel in a straight line and cuts the fourth meander short. On the other hand, on the flood plain east side, the flow sandwiched between the main channel and the zone immediately along the west bund displays a sharper westward redirection, heading for the constriction at section 6, Fig. 6.12.

At Cross-Section 1, Fig.6.65, the velocity core is located on the right bank in the main channel and the flow appears to be rotating in the clockwise direction, Fig. 6.69. However, this is mostly the result of a two-layer flow, with the flow in the main channel following the main channel geometry and the overbank flow cutting across the bend in a straight line. This is suggested on the angle plot, Fig. 6.67, which presents a horizontal gradient of the angle, increasing with depth. On the top left corner the negative angle indicates that the flow on the left flood plain is simply cutting across the corner. This generates a relatively strong recirculation, estimated to be equivalent to 25% of the overall velocity maximum.

At section 2, at the entrance of the second bend, the main velocity filament (0.6 m/s) is located along the inner bank. Looking from the inner bank towards the outer bank, one notices that the flow is progressively redirected in the direction normal to the section, shifting from positive (15°) to small negative angles. A slight anticlockwise rotating motion is probably induced by the centrifugal effect at the bend and part of the flood plain flow coming across the right bank, Fig. 6.69.

At Section 3, Fig. 6.65, the flow in the main channel is mostly travelling in a direction normal to the section. The velocity maximum in the main channel (0.5 m/s) is located in its centre. As expected, the velocity peak (0.9 m/s) is located along the right embankment on the flood plain. This is the result of the water being ejected from the main channel and the flow crossing from the left flood plain at section 1. At this location the flow is travelling in a direction parallel to the tangent to the embankment (15°), Fig. 6.67. Both velocity and angle decrease towards the main channel, as the flow is slowed down and re-aligned. However, the lateral velocity in the main channel is as strong as it was at section 2, Fig. 6.69, yet it corresponds to a stronger intensity (25%) since the overall velocity is slower at section 3 due to the expansion of the cross section.

Further downstream, at section 4, the flow is shifted towards the outer bank. It is rotating in an anticlockwise direction, with an intensity equivalent to 25% of the cross-section main velocity. This is what would happen in an inbank flow situation, yet it is further accentuated here by the contribution of the flow coming from the right flood plain, as indicated on the angle plot, Fig. 6.67. The vertical velocity results in the calculation of a spurious velocity along the bottom part of the left bank (0.7 m/s). This sequence of events is similar to what was simulated at section 3 in the models of the River Severn.

Section 5, Fig. 6.66, presents characteristics similar to the cross-over region in the Flood Channel Facility (FCF). The depth ratio is equal to 20%, and the flood plain flow is travelling at an angle of approximately 60° with respect to the normal to the section. This results in a velocity profile exhibiting very similar characteristics to the observations made at the cross-over in the FCF. The lateral flow contributes to shift the velocity maximum (1.1 m/s) to the left and stretch the main isovels along the left bank. However, the flood plain velocity is hardly 50% of the main channel's which reduces its impact on the flow structure in the main channel compared to the FCF. The flood plain flow is rapidly re-directed downstream, Fig. 6.68, which results in a secondary cell that is weaker than expected.

At Section 6, the main velocity filament is flowing against the left bank towards which it is shifted by the lateral flow coming from the flood plain on the right. The shape of the isovels speaks for the existence of a strong anti-clockwise recirculation, which is confirmed by the angle plot, Fig. 6.68. Nevertheless the flow interaction at this section is not as simple as it may appear. The angle plot, Fig. 6.68, also suggests that the water flowing on the left flood plain in a direction parallel to the south embankment interferes with the flow structure in the main channel, which rotates in a direction opposite to that of the shear generated by the left flood plain flow. This contributes to reduce the anticlockwise helical motion, and probably dampens the shear drive effect from the flow coming from the central flood plain.

Past the last bend, the flow behaves more like an inbank flow, section 7, Fig. 6.66. The velocity core (1.0 m/s) is progressively moving away from the inner side of the bend, however, a relatively fast flow is present on the left flood plain. It results from the flow that crossed over the main channel at section 6 and proceeded to cut across the bend, Fig. 6.68. It affects the lateral distribution of velocities in the top part of the channel.

At section 8, the flow is fairly simple and travelling downwards mostly. However, the angle plot, Fig. 6.68, shows that the flow is still rotating in an anticlockwise direction. The left flood plain flow travels towards the main channel and acts against the rotation in the main channel. The intensity of the recirculation at section 8 is estimated to be about 5% of the main channel velocity.

### 6.5.2 Fully-3D Model of the Ribble using CFX

#### 6.5.2.1 Determination of the Wall Roughness

In the absence of detailed information regarding the nature of the bed material in the main channel, boundary roughness was evaluated qualitatively from the knowledge of the flow properties, roughness tables available in textbooks such as Chow (1959), and the calibration work previously conducted with TELEMAC.

During the winter months the Ribble central flood plain is relatively smooth and covered with short pasture grass. Consequently a low Manning's n of 0.030 seems adequate for the flood plain (Chow, 1959). It corresponds to a Chézy C of 40 $m^{1/2}/s$ or a roughness height, $k_s$, of 0.120 m. In the main channel calibration work carried out with TELEMAC yields a Chézy C in the region of 50 to 55 $m^{1/2}/s$. It is equivalent to a $k_s$ value of 0.080 m according to equation (6.2). The walls of the main channel are believed to be covered with a sandy material finer than the small gravel found in the Severn. Hence a value of 0.080 appears to be consistent with the values determined for the Severn.

#### 6.5.2.2 Sensitivity Analysis and Calibration

As for the model of the Severn, most of the sensitivity analysis focuses on adjusting the roughness height to obtain the smallest pressure field on the surface-lid of the CFX model. The impact of roughness on the velocity field at the observed cross-sections is also monitored.

Having estimated an approximate range of values for the roughness coefficient, a first simulation run with a roughness height of 0.120 m on the flood plain and 0.080 m in the main channel is used as a reference. It produces a reasonable match with the TELEMAC free-surface predictions although differences equivalent to 4.0 cm of hydrostatic pressure exist between the two models close to the downstream end, Fig. 6.71. This however represents an error inferior to 1% of the total water depth at this location. Interestingly the pressure field on the lid, Fig. 6.72, shows the pressure build up on the outside of the bends. Similarly a lower pressure field is present in the inner bends, and the model therefore captures well the inclination of the free surface across the section of the river at the bends (Wright and Morvan, 2000). However, there is a zone of spurious low pressure at one point in the last bend (-1,400 Pa) which could not be eliminated. One also notices another region of low pressure where the water is ejected on the flood plain, past the first bend. This is equivalent to the contraction noticed at the cross over in the FCF. Such dynamic effect is not included in the lid and therefore results in a "depression" equivalent to about 10% of the flood plain water depth here. This makes it a bordering case regarding the validity of the lid assumption. However, the pressure builds up rapidly further down on the flood plain and the difference in the computed pressure on the lid falls within 5% of the total flow depth in terms of water head. In comparison with Sinha *et al.* (1996), such localised pressure variation remains inferior to what they obtained in part of their model of the 4-km reach of the Columbia River.

A second simulation is run using rougher conditions for both channel and flood plain, using roughness height values of 0.120 m and 0.240 m respectively. Little difference is visible between the first and second simulations. The pressure gradient tilts downwards slightly when roughness increases, as observed in the FCF and River Severn models. This means that the solution worsens in the lower end of the reach and indicates that this range of values is not as good as the previous one.

Other attempts with lower roughness values did not appear to bear significant impact on the pressure field. They contributed to show that the model is very insensitive to large changes in roughness.

The calibration set up used in the first simulation results in a pressure field on the lid that is consistent with the free surface calculated with TELEMAC. This seems logical in a way since the CFX model was built from the results obtained with TELEMAC.

6.5.2.3 Predicted Velocity Field and Flow Mechanisms

In the absence of validation data for the River Ribble, the numerical results will simply be discussed qualitatively with respect to the FCF experiment. A comparison with TELEMAC's predictions will be made in section 6.5.4.

A peak flow discharge of approximately 98 $m^3/s$ is simulated here. A plan view of the surface flow is shown on Fig. 6.73. At the first bend the flow travels across the left flood plain and the main channel onto the central flood plain. Further downstream, the flow splits due to the earth protrusion running along the main channel right bank at the second bend. Along this earth protrusion, the velocity is very low. Most of the water on the flood plain comes straight from the inlet section. Between the first and second bends there seems to be less lateral flow contribution from the main channel on to the flood plain than predicted in the TELEMAC model. Between the third and fourth bends the impact of the fast flood plain flow on the main channel's results in the main channel flow shifting towards the outer bank. This is increased at the fourth bend where the flow coming from the central flood plain cuts across the main channel and the bend, to dive in the straighter part of the reach further downstream.

At cross-section 1, Fig. 6.74, the above observations result in a maximum velocity of 0.8 m/s flowing in the centre of the main channel. In the upper part of the channel the angle plot shows that the flow travels at a positive angle of about 10°-20° with respect to the normal, Fig. 6.76. In the main channel the horizontal gradient of the angle and the maximum angle of 60° at the bottom suggest that the flow in the main channel dives as it hits the outer bank, which generates a clockwise rotating structure similar to that observed in the FCF. This phenomenon is mostly driven by the flow contribution from the left flood plain cutting across the main channel, and results in a strong recirculation equal to 20% of the maximum velocity, Fig.6.78, in the central part of the channel.

At cross-section 2, Fig.6.74, the velocity core travels along the channel inner bend and reaches a maximum of 0.5 m/s. The angle plot shows that inertia makes the flow drift slightly towards the outer bend in the main channel, and that there is a sharp change in direction as the flow jump onto the left narrow flood plain. This generates a very weak recirculation to the left, Fig. 6.78.

Further downstream at section 3, the velocity maximum in the main channel (0.5 m/s) is located in its centre. A maximum isovel of 0.9 m/s is pictured along the right embankment. The velocity reduces away from the west embankment towards the main channel. The velocity in the channel is travelling in a direction normal to the section, while the velocity on the flood plain travels at an angle to the left. This angle increases away from the west embankment to reach 20° where flood plain and main channel meet. This lateral inflow into the main channel induces a counter-clockwise rotation equal to 20% of the maximum inbank velocity (0.5 m/s).

At cross-section 4, Fig.6.74, the velocity maximum (0.5 m/s) is located towards the right of the channel in its lower part. As shown on the angle plot, Fig. 6.76, a flood plain inflow travelling at an angle of 40-50° shifts the isovels to the left in the upper part of the main channel. This results in a complex flow situation with part of the flow turning to the right in the main channel, and part of it crossing over from the flood plain. It is therefore difficult to determine a main motion pattern from a simple recirculation balance in the centre of the channel, Fig. 6.78.

Cross-section 5 constitutes a situation similar to that of section 4 in the Severn, where a strong lateral inflow crosses over from the flood plain and is forced to turn to the right. This situation results in the main isovel being squeezed against the left bank, which produces a recirculation plot, Fig. 6.79, reminiscent of the observations carried out at the cross-over in the FCF. This recirculation is equal to 25% of the local velocity maximum at the surface, Fig. 6.80. Most of the flow in the section travels towards the left bank, which suggests that there might not be any fully formed helical motion in the main channel yet.

Cross-section 6 displays similar velocity and angle profiles to that of section 5, albeit with a stronger impact from the flood plain inflow. It is also very similar to the results plotted for the cross-over in the FCF work presented in Chapter 5. Additionally, Fig. 6.77 seems to indicate that the flow separates along the right hand side and that a recirculation is formed in the plan view, Fig. 6.82(a). Because the channel section is reduced at section 6, the flow is accelerated and a maximum velocity of 1.0 m/s is reached along the top part of the left bank. On the left bank the angle is negative, probably due to the flow running parallel to the outer embankment on the narrow left flood plain, Fig.6.77. The angle plot also suggests that there is a helical motion in the main channel with velocities heading from the "right to the left" to the "left to the right" as depth increases. Its strength must be slightly hindered by the lateral flow coming from the left. The recirculation is at its strongest at section 6, Fig. 6.80, however. The inbank water is stirred within the cross-sectional plane with a maximum velocity of 0.2 m/s, a similar magnitude to that predicted for the Severn in a similar situation.

Section 7 being in the immediate continuity of section 6, the velocity features are similar, except that there is no lateral inflow from the right bank. This means that the isovels are "vertical" in the main channel. The inbank velocity maximum reaches 0.9 m/s while a faster velocity equal to 1.0 m/s can be seen on the left flood plain due to the flow cutting across bend 4, Fig. 6.73. The latter flow starts to re-join the main channel at section 7, as suggested in Fig. 6.77. As a result the recirculation in the main channel centre, Fig.6.79, is weaker than at section 6 (10-15% of the maximum velocity).

At section 8, the flow that was travelling on the left flood plain has dived into the main channel. The shape and location of the main 1.0-m/s isovel clearly underlines this: It has shifted to the channel right hand side. This is confirmed on the angle plot where the region of negative angles predicted in the top left corner in the main channel accounts for the flow coming from the left flood plain. This contributes to the cancellation of the helical motion, Fig. 6.79, which intensity is reduced to about 3% to 5% of the channel overall velocity.

6.5.2.4 Numerical Tracers

Numerical tracer results are worthy of a brief discussion as they reinforce some of the descriptions made earlier regarding the motion of the flow. For example they clearly help identify zones of recirculation on the left bank between bends 1 and 2, Fig. 6.80(a), and in the vicinity of cross-section 6, Fig. 6.81(a). Downstream of section 4, the inbank flow is made to rotate, which is due to the flood plain flow crossing over the main channel, as was underlined in the previous analysis. At the fourth bend the flow in the main channel remains in the confines of the channel where it is strongly stirred, Fig. 6.81(b), by the flood plain flow cutting across in a straight line, Fig. 6.82(b).

This type of result is certainly on par with similar experimental results obtained in Chapter 5, e.g. Fig. 5.53 and 5.54. It offers great possibilities to visualise complex flow motions and make use of three-dimensional data.

6.5.2.5 Predicted Bed Shear Stresses

Bed shear stresses were calculated at all eight cross-sections, using the turbulence kinetic energy results computed during the simulation. They are plotted on Figs. 6.83 to 6.90.

At cross-sections 1 the bed shear stress curve bears the results of the flow cutting across the first bend. A maximum shear stress of 1.6 N/m$^2$ is calculated on the left flood plain. At cross-section 2, the velocity main filament leaning against the inner bank induces an increased shear stress reaching 1.0 N/m$^2$ locally. At cross-section 3, the same effect is visible at the foot of the right embankment due to the flow travelling in a straight line from the inlet. At section 6, the average shear stress is about 1.3 N/m$^2$ in the main channel, with a peak of 2.2 N/m$^2$ at the bottom of the right bank. It seems quite surprising not to witness a higher value of shear stress at this section. It could be that the flow running parallel to the south embankment on the left flood plain limits the impact of secondary motion on the left wall in the main channel. At cross-sections 5 and 6, the channel larger width-to-depth ratio could also account for the existence of lower vertical velocities at the banks and a weaker cross-sectional recirculation.

In general the level of bed shear stress in the model of the River Ribble reach is lower and more uniform than in the Severn. This may reflect the existence of weaker secondary currents (lower vertical velocities) and of smoother boundaries in the main channel, and is consistent with the absence of significant marks of erosion on the banks.

#### 6.5.2.6 Turbulence Anisotropy in Flooded Rivers

Although previous tests conducted in the FCF have indicated that turbulence anisotropy played little role in the determination of the flow structures in flooded meandering channels, a simplified version of the Reynolds stress model (RSM) presented in Chapter 2 was implemented for the CFX model of the Ribble.

When comparisons between the results obtained with the $k$-$\varepsilon$ model and the LRR-RSM were made variations appeared, at cross-sections 1, 5, 6 and 8, Fig. 6.91; yet they were judged to have limited impact on the interpretation of the overall results for the resolution level considered by the author. Most of the flow features predicted by both models are visibly dominated by the interaction of flows travelling in different directions. This is in line with the conclusions presented in Chapter 5. Pure turbulence contributions are much weaker and over-shadowed by the effects of the shear flow effects. It is therefore clear that a more complete model such as the RSM does not result in better predictions. On the contrary, it is more of a "liability" in the sense that it is more difficult to implement in such complex situation. These remarks also explain why TELEMAC output compares reasonably well with that of CFX here again.

### 6.5.3 Models of the River Ribble: Conclusions

As with the models of the River Severn the results presented above display features that are reminiscent of the flow investigated in the FCF. Although fully three-dimensional the flow is fairly simple and mostly driven by the interaction between inbank and overbank flows travelling in different directions. Such interaction results in strong secondary currents that are rotating in directions opposite to those observed at river bends in inbank flow situations, because the drive of the flood plain flow is more important. This implies that the role of turbulence transport is minimal, which was confirmed here in a comparison between viscosity and second order closure models.

Both codes produce similar predictions, at the exception of sections 5 and 6 where a marked difference exits. TELEMAC predicts that the flow at section 6 is already re-aligned with the main channel, whereas CFX indicates that this section is where the flood plain flow crosses over the main channel and generates the strongest secondary current. Such difference could have repercussions on the interpretation of the overall channel dynamics. This difference could be due to the failure of the mixing-length model to calculate local recirculations. Moreover it seems that, in the CFX model, the impact of the lateral inflow from the central flood plain is stronger than in TELEMAC, particularly at cross-sections 5 and 6. Part of this difference could be a matter of calibration, e.g. for the flood plain roughness, as illustrated in section 6.5.1.2. As usual CFX predicts a slower flow through the domain, and at cross-section 5 in particular, which adds to the discrepancy. Regarding the flow direction, similar comments apply: The flow pattern appears to be globally the same, yet local differences, e.g. at cross-section 6 again, result in an overall difference that can reach up to 25° at the bed. In turn these differences impact on the prediction of secondary currents within the cross-sectional plane, as evidenced at sections 4 and 6. It is however difficult to determine which model is more representative of reality with certainty. The lack of field information entails that local uncertainties in the flow predicted by the models cannot be resolved.

Once again the domination of advection effects explains the good comparison between the two models, although vertical velocities at the wall are poorly calculated with TELEMAC due to the nature of its formulation. TELEMAC models prove very useful to determine the position of the free surface and are more flexible than CFX in modelling the topography.

These results obtained with these models support the conclusions reached earlier from the analysis of the River Severn model outputs regarding:

(i) the nature of the flow mechanisms operating in flooded meandering channels;

(ii) the existence of similarities between the FCF and natural channels;

(iii) the low impact of turbulence on the helical flow in the meanders.

## 6.6 SUMMARY, DISCUSSION AND CONCLUSIONS

This chapter has presented the application of commercial Computational Fluid Dynamics (CFD) to natural river channels to simulate flood flows.

### 6.6.1 Constraints and Limitations

The main difficulty of applying such technique to natural channels lies in the representation of the geometry in three dimensions, especially with codes such as CFX. There is a clear antinomy between the requirements of the code for a well-behaved geometry that can easily be described mathematically and the nature of the river channels, which is highly irregular. Additionally the quality of the topographical sampling is bound to determine how accurate and smooth the modelled domain can be. This affects the quality of the structured numerical grid and, ultimately, that of the solution, especially when fine grid resolutions are implemented. Furthermore scale and roughness contributes to stretch further the gap between the resolution that can be achieved and the need for a fine grid at the walls, even if this should not constitute a fundamental obstacle to the use of CFD in river engineering for the prediction of the main flow pattern. Finally the construction of the rigid lid using external data in CFX is a shortcoming of this formulation since it makes it dependent on either field, laboratory or TELEMAC free surface data.

Calibration and roughness are also major constraints in the implementation of CFD in natural rivers. Roughness in particular is an issue that would require further investigation, as the link between the numerical calibration values and the knowledge acquired from laboratory or field work remains poorly understood. Although the roughness values used in the two CFX models have been related to some experimental formulations in this study, this has been done with the knowledge that these formulations have either been applied outside their usual range of validity, or – although they allegedly relate bed particle size and roughness – include a wide range of energy loss mechanisms. Carling (2001) suggested to relate the scalar values used for $k_s$ in CFD models to surface types and forms found in the field. This would permit the association of an overall ground roughness to a corresponding momentum loss coefficient in the models, as done by Acrement and Schneider (1989) for Manning's n values for example. In the process the grain-roughness relationship would be put aside. Because of the level of complexity and uncertainty behind the definition of roughness, this might be an ambitious and rather uncertain approach, especially since there are other "numerical" mechanisms associated with energy losses (e.g. grid resolution, turbulence models) which would probably interfere with the construction of such relationship. Another idea that is being developed between Glasgow and Lancaster Universities would be to investigate, experimentally and numerically, the effects of separate physical mechanisms (grain, bedform, turbulence) and numerical factors (grid resolution, law of the wall) on the momentum loss coefficient, extending in a way the work of Clifford *et al.* (1992). This would be carried out in a large-scale flume, to optimise our control of the varied parameters.

Regarding the accuracy of the models, calibration is a major issue in the sense that different set of parameters might lead to similarly identical flow features when compared with sparse data (especially for free-surface codes). Indeed the more complex the models, the more parameters they incorporate, and therefore the more difficult it is to conduct a perfect calibration, especially considering the level of uncertainty associated with collecting data in rivers. For example, it has been shown that for seemingly identical water profiles, difference of 10% could easily be found in the prediction of velocities in TELEMAC, depending on the choice of roughness values. If turbulent parameters had been altered further difference might have occurred. When cumulated with an average level of error in velocity measurements in the field (10%) and additional topographical errors, this illustrates how difficult it can be to optimise distributed models and compare them objectively with measurements. It should be noted that, at the opposite end of the spectrum, the lack of sensitivity of the rigid lid models to roughness in particular is as much of a handicap, as it greatly constrains their construction.

### 6.6.2 Quality of the Predictions

The results obtained for both river reaches indicate that, as in the FCF, the flow is geometry-driven and advection dominated. The interaction between inbank and overbank flows results in the emergence of a strong helical motion within the main channel walls, for which most mechanisms are identical to that in the FCF. Comparison between field data and the models of the Severn in the region of the cross-over also confirms that the models are able to represent the river main flow features faithfully, e.g. in terms of the shape, location and magnitude of the isovels. This implies that they could be used in engineering design to determine the main three-dimensional flow features, despite the comparison between scattered point data and the model predictions revealing errors of 20% to 30% in velocity in places.

Analysis has shown that several factors could be responsible for the scatter, in particular the mesh resolution at the walls, the choice of friction factors and uncertainty in the field data. The scarcity of field information implies that a full validation cannot be made and it is difficult to form an opinion based on rare point data. From a CFD point of view such level of uncertainty is unsatisfactory. Yet, within the frame of river engineering, the comparison between predicted and measured isovels at sections 4 and 5 is encouraging because the attainable level of accuracy in river engineering is very different from that in other industrial field of application of CFD. This is true for both field and numerical work. The interested reader is referred to Sinha *et al.* (1996, Figs. 5.1 to 5.3 pp. 141 to 143) for example, where similar level of errors between data and model were also found locally. Such difference should not prevent engineers from using the technique, provided they are aware that it exists.

Circumstantial evidence was used to assess the prediction of bed shear stress in the Severn. Although there is some doubt regarding the numerical accuracy of the predicted values, the pattern of the bed shear stress across the domain appeared to be consistent with the distribution of bed sediments and evidence of bank erosion. From the model the author would have been able to identify the second meander as a critical region. This reinforces the idea that CFD could be used to understand and quantify the main flow features in complex river flow situations. Constructing a numerical model might in the end be cheaper and more flexible to investigate various scenarios, than using a scale model.

Although both codes have yielded similar predictions for the models of the Severn and Ribble reaches, local differences in velocity magnitude and direction have sometime resulted in different predictions at the same location. Differences in the velocity magnitude in the main channel can be attributed to pressure effects and the absence of recirculating velocities in the TELEMAC models is probably due to the mixing-length model. However, difference in the direction on the flood plain velocity could have a lot to do with friction effects and the representation of the geometry, which illustrate how difficult the construction of an accurate model can be.

### 6.6.3 Quasi-3D vs. Fully-3D?

In general both numerical codes used in the present work give similar predictions, although TELEMAC's description of the flow is more limited and often dominated by the layered approach. Its good performance is clearly related to the fact that the main flow features are driven by horizontal shear acting between the layers. The representation of vertical flow features is impossible, which impairs the code's ability to predict erosion effects in three-dimensional flows. Whenever strong vertical currents exist they are inaccurately calculated. This has unquantifiable effect on local mass conservation. Additionally, there is evidence from the work of Stansby and Zhou (1998) that a hydrostatic pressure results in slightly faster flows in trench flows or flows over a hump. From a physical point of view CFX is better at accounting for fully three-dimensional flows. In particular, the calculation of pressure and vertical motion ensures that their effects on the free surface, on bed shear stress (Lane *et al.*, 1999) and sediment entrainment (Fukuoka and Wanatabe, 2000), or on hydrodynamics are more accurately calculated (Stansby and Zhou, 1998). These are all essential features in engineering design.

On the other hand, the calculation of the free surface in TELEMAC is a major advantage for several problems related to fluctuating water levels. This code is certainly better equipped than CFX for the discretization of river channel morphology, and large-scale problems. Models using TELEMAC are also more readily constructed and calibrated. The choice of either type of code should be made from the perspective of the usage for which the model is destined. The decision might come down to the balance between scale (which favours TELEMAC) and the level of detail and accuracy sought in the predictions (CFX). However, TELEMAC cannot be recommended for detailed three-dimensional predictions in rivers for inbank flows where pressure and turbulence driven effects are significant.

### 6.6.4 Conclusions

Computational Fluid Dynamics (CFD) works reasonably well in natural river channels. Its application is interesting to identify the characteristic features of the flow mechanisms as well as their relative magnitude, especially when the flow is turbulent, three-dimensional and that significant spatial variations exist within the domain. Furthermore evidence presented in this chapter suggests that reasonably good numerical predictions can be made and used in river engineering design for the velocity and the distribution of bed shear stress when using a fully three-dimensional approach. Finally, the models display results that are very similar to the observations and calculation shown in Chapter 5 for the Flood Channel Facility (FCF).

One of the main difficulties for the application of CFD to natural rivers lays in the construction and discretization of the domain. This implies that this technique is relatively expensive to use and that predictions at the walls are poor, which would limit the use of CFD for sediment transport in full-scale problems, unless unstructured grids and more adequate formulations for the law of the wall become more readily available. As illustrated in several instances in this study, sensitivity analysis to the boundary conditions and model parameters is also an important, yet uncertain, issue. With free-surface models, sensitivity to these conditions has been found to lead to equifinality issues, i.e. multiple solutions. On the other hand, the lack of sensitivity observed in rigid lid models also makes them very difficult to construct and adjust. In general, because of the amount of information required, and subsequently processed, by these models it can be difficult to obtain a unique, perfect solution. This illustrates that there is room for progress, and that users should therefore remain critical of their predictions. They should be aware that the latter are only as good and accurate as the model's input data, constitutive relationships and algorithms permit.

## 6.7 REFERENCES FOR CHAPTER 6

1. Acrement, G.J., Schneider, V.R., (1989), "Guide for Selecting Manning's Roughness Coefficients for Natural Channels and Flood Plains", U.S. Geological Survey Water-Supply Paper 2339.
2. Baker, A.J., Orzechowski, J.A., (1983), "An Interaction Algorithm for Three-Dimensional Turbulent Subsonic Aerodynamic Juncture Region Flow", AIAA Journal, Vol. 21, No. 4, pp. 524
3. Bates, P.D., Horritt, M.S., Hervouet, J.M., (1998), "Investigating Finite Element predictions of flood Plain Inundation using Fractal Generated Topography", Hydr. Proces., Vol. 12, pp. 1257-1277.
4. Bettess, R., (2001), *personal communication.*
5. Carling, P.C., (2001), *personal communication.*
6. Chow, V.T., (1959), "Open Channel Hydraulics", Mc-Graw-Hill International Edition, Civil Engineering Series.
7. Clifford, N.J., Robert, A., Richards, K.S., (1992), "Estimation of Flow Resistance in Gravel-Bedded Rivers: A Physical Explanation of the Multiplier of Roughness Length", Earth Surface Processes and Landforms, Vol. 17, pp. 111-126.
8. Easom, G., (2000), "Improved Turbulence Models for Computational wind Engineering", Unpublished PhD Thesis, The University of Nottingham, UK.
9. French, R.H., (1985), "Open-Channel Hydraulics", McGraw-Hill International Edition, Civil Engineering Series.

10. Fukuoka, S., Wanatabe, A., (2000), "Numerical Analysis on Three-Dimensional Flow and Bed Topography in a Compound Meandering Channel", 4th Int. Conf. Hydroinformatics 2000, Iowa Institute of Hydraulic Research, IAHR CD-ROM.
11. Hankin, B.G., Hardy, R., Kettle,H., Beven, K.J., (2001), "Using CFD in a GLUE Framework to Model the Flow and Dispersion , Earth Surface Process. and Landforms (to appear).
12. Horritt, M.S., (2000), "Calibration of a Two-Dimensional Finite Element Flood Flow Model Using Satellite Radar Imagery", Water Res. Res., Vol. 36, No. 11, pp. 3279-3291.
13. Janin, J.M., Marcos, F., Denot, T., (1997a), "Code TELEMAC-3D Version 2.2 – Note Théorique", Report HE-42/97/049/B, EDF-DER, LNH Chatou, France (In French).
14. Janin, J.M., David, E., Denot, T., (1997b), "TELEMAC-3D Version 2.2 – User's manual", Report HE-42/97/051/B, EDF-DER, LNH Chatou, France.
15. Lane, S.N., Bradbrook, K.F., Richards, K.S., Biron, P.A., Roy, A.G., (1999), "The application of Computational Fluid Dynamics to Natural River Channels: Three-Dimensional versus Two-Dimensional Approaches", Geomorphology, Vol. 29, pp. 1-20.
16. Manson, J. R., (1994), "The Development of Predictive Procedure for Localised Three Dimensional River Flows" PhD Thesis, The University of Glasgow.
17. Mesehle, E.A., Sotiropoulos, F., (2000) "Three-dimensional Numerical Model for open-Channels with Free-Surface Variations", J. Hydr. Res., Vol. 38, No. 2, pp. 115-121.
18. Peltier, E., LeNormant, C., Teisson, C., Malchereck, A., Markofsky, M., Zielke, W., Cornelisse,J., Molinaro, P., Corti, S., Greco, G., (1996), "Three-Dimensional Numerical modelling of Cohesive Sediment Transport Processes in Estuarine Environments – Final report to the EC Contract MAS2-CT92-0013", Report HE-42/97/049/B, EDF-DER, LNH Chatou, France.
19. Shyy, W., Vu, T.C., (1991), "On the Adoption of Velocity Variable and Grid System for Fluid Flow Computations in Curvilinear Coordinates", J. Comp. Phys., Vol. 52, pp. 82-105.

20. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.
21. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1996), "Numerical Model Studies for Fish Diversion at Wanapum/Priest Rapids Development – Part I: Three-Dimensional Numerical model for Turbulent Flows through Wanapum Dam Tailrace Reach", IIHR Limited Distribution Report No. 250, Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa, USA.
22. Stansby, P.K., Zhou, J.G., (1998), "Shallow-Water Non-Hydrostatic Pressure: 2D Vertical Plane Problems, Int. J. Numer. Meth. Fluids, Vol. 28, pp.541-563.
23. Versteeg, H.K., Malalasekera, W., (1995), "An Introduction to Computational Fluid Dynamics – The Finite Volume Method", Longman.
24. Wu, W., Rodi, W., Wenka, T., (2000), "Three-Dimensional Numerical Modelling of Flow and Sediment Transport in Open Channels", J. Hydr. Eng., Vol. 126, No. 1, pp.4-15.
25. Ward, R.C., (1981), "River Systems and River Regimes" in "British Rivers", J. Lwin (Ed.), George Allen and Unwin, London, pp.1-33.

# Chapter 7:

# Conclusions and Recommendations

# Chapter 7:

# Conclusions and Recommendations

## 7.1 INTRODUCTION

This work has presented the first investigation into the use of Computational Fluid Dynamics (CFD) for the calculation of river flood flow detailed mechanisms in natural channels. It also constitutes one of the few research efforts published to date in the field of full scale river modelling in three dimensions; previously only Sinha *et al.* (1998), Hodskinson and Ferguson (1998) or Lane *et al.* (1999) have preceded the author.

Two commercial codes have been used in the present work, with reasonable success: One a specialised quasi-3D hydraulics code making use of a hydrostatic assumption to present layers of two-dimensional solutions (TELEMAC), the other, a general CFD code with a more mechanical engineering background (CFX).

This work had three main objectives:

(i) the application and assessment of commercially available advanced CFD techniques to the simulation of natural river flows;

(ii) the investigation of the flow mechanisms at laboratory and full-scale in two-stage channels during flood events, and the establishment of their similarities, using CFD;

(iii) the presentation of guidelines regarding the important requirements for a proper implementation of CFD to river engineering problems.

The following section will review each aspect of these three objectives in reverse order and present the conclusions reached by the author during the course of his work.

## 7.2 CONCLUSIONS

### 7.2.1 Model Construction

- Spatial Discretization: One of the most challenging problems with numerical modelling of flow through natural river reaches stems from the complexity of the geometry and its disruptive effects on the numerical solution. In this context the use of a structured grid (although multi-block) in CFX has proved quite difficult and entailed a lot of manual mending and smoothing of the boundaries. On the other hand the unstructured approach available in TELEMAC offered more flexibility and appears to be more appropriate to model river channels.

- Grid Resolution: The use of a proper spatial discretization is essential to a good representation of the flow features in regions of sharp gradients of the variables. In the case of overbank flows, the resolution has to be enhanced at the flow interface between the flood plain and the main channel and at the banks. For similar reasons it is desirable to have a fine grid spacing at the walls in order to capture the velocity profile until the boundary, as shown in Chapter 5. This is not always feasible, particularly in natural channels due to their scale, Chapter 6. The present work, however, indicates that CFD techniques provide an adequate simulation of complex three-dimensional flows in rivers if one excludes the solution of the near-wall flow. With TELEMAC a fine resolution is also desirable at the interface between dry and wet cells to allow for a good representation of the free surface.

- Mass Conservation: Another feature to investigate with TELEMAC is mass-conservation, since it is well known that finite element techniques are not mass-conservative, and that the grid not only impacts on the continuity equation, but also on the spatial distribution of mass within the domain. In the present investigation the use of very fine grids guaranteed mass conservation, and ensured mesh-independent solutions.

- Numerical schemes: The order of accuracy of the numerical scheme has been shown to have little impact on the solution of natural river flows, Chapter 6. The advection-dominated flow was well represented by a first order scheme, as was the case in recent contributions by Sinha *et al.* (1998), Hodskinson and Ferguson (1998) or Lane *et al.*, (1999), and did not exhibit excessive false diffusion despite the use of relatively coarse grids. In the model of the Flood Channel Facility (FCF) the author used a third order accurate scheme in order to eliminate traces of numerical diffusion, since a much finer resolution of the velocity profiles was desired in this particular case. In general, higher order schemes are recommended in CFD (Easom, 2000), yet their usefulness really depends on the level of false diffusion in the solution and the resolution sought by the end user.

- Inlet Boundary Conditions: Tests conducted in the present study, Chapter 5, have demonstrated the limited dependence of the flow solution in the main channel to the boundary conditions at the inlet. Another way to view this is that the synthetic velocity and turbulence profiles reviewed in Chapters 4 and 6 provide a sufficiently accurate boundary condition. This is an important conclusion since field data, other than an estimate of the discharge and the position of the free surface, are rarely available to model flood events in practise.

- Wall Boundary Condition and Calibration: As reported by various other researchers (Bradbrook *et al.*, 1998; Ma Lin, 2000) rigid-lid models are relatively insensitive to their boundary conditions, in particular at the walls. Additionally, the author has noticed that whenever high or low boundary roughness values (compared to the estimated value) are used, models can experience difficulties in converging properly. Referring to Abbott (1989), this was attributed to the fact that a rigid lid model is constructed to be consistent, i.e. the different boundary conditions (inlet flow, lid position, wall roughness) have to be consistent with each other in order to obtain satisfactory convergence. A direct implication is that, in such models, calibration is not so much the process of tuning the wall roughness to the laboratory and field data, but the process of making all boundary conditions consistent with each other for the simulation of a given event. This makes the model construction much more intricate since there is little freedom for post-construction adjustment.

- Law of the Wall and Calibration: It is important that rough wall boundary conditions are implemented in natural rivers, especially when using mechanical engineering flow solvers. The author reviewed four laws in Chapter 4, two empirical and two theoretical. However, there is some uncertainty regarding the role of such laws when the grid is coarse, especially since the solution seems to exhibit little variation to its calibration in rigid-lid models. Indeed CFX seemed more sensitive to variations in roughness with smooth walls and fine grids, Chapter 5, than with rough wall conditions in natural channels, Chapter 6. Precise calibration is also very difficult with free-surface codes although they are more sensitive to variations in roughness and seem more adjustable. Complex variations in the water elevation and velocity profiles could make this task a never-ending process due to the distributed nature of the predictions. It is also important to recognise that it merely constitutes a sensitivity analysis and that other factors such as turbulence, grid resolution and numerical accuracy and uncertainty in the data would also need to be addressed.

- Wall Roughness: The author has noted that there is an important lack of information regarding the parameterisation of laws of the wall via a roughness height value, especially as the bed is made of coarse gravel or covered in vegetation. This adds to the uncertainty regarding the application of the law of the wall and requires further investigation.

- Free-Surface Modelling: The modelling of the free surface as a rigid lid using experimental and field data works well in CFX. This, however, is a shortcoming of the code present formulation since it makes it dependent on laboratory or field data. Ways to address these issues are presented in the context of future work in section 7.3. On the other hand the free-surface modelling in TELEMAC is satisfactory since it allows for a deformable mesh. However, it also implies that simulation of vertical velocities is poor and that the dynamic effects of pressure are not reproduced, both of which can have strong implications in river engineering.

- Turbulence Models: Simple turbulence models were found to be adequate to predict the flow satisfactorily in meandering two-stage channels. However, the good performance of such models in this particular case cannot be generalised to all river reaches, especially those with strong meanders and inbank flows where turbulence stress-driven cells are of interest.

- Numerical Solvers: Following recommendations made in recent publications (Sinha *et al., 1998, Mesehle* and Sotiropoulos, 2000) a multigrid algorithm was implemented in CFX to resolve the open-channel flow problems. Stone's method, an improved ADI algorithm, was also used in Chapter 5 for comparison. It compared well until the mesh resolution became very fine. Multigrid capacity to deal with the different components of the numerical error then appeared to be an essential characteristic to reach convergence efficiently. This characteristic becomes particularly important where the grid is complex and made of various resolution levels in different parts of the domain. Moreover multigrid would appear to alleviate part of the constraints imposed by the geometry upon the numerical solution when using structured grids.

### 7.2.2 Flow Mechanics

- The flow in meandering channels during flood events is highly three-dimensional, advection-dominated and mostly geometry-driven. The interaction of the main channel geometry with fast flow crossing over the channel results in strong rotating structures within the walls of the main channel, which intensity is mostly related to the angle between the two flows, their relative depth and relative velocity. Within a meandering reach this creates a rotating flow pattern that is opposite and much stronger than that observed for inbank flow situation, consequently with the potential to destabilise the channel morphology. In the case of the FCF secondary cells of intensity close to 50% were observed, whereas both the Severn and Ribble models predicted lateral recirculation of intensity equal to 30% of the cross-sectional maximum velocity magnitude.

- This is important in design, especially where structures are constructed on the riverbanks and on the flood plains. The model of the FCF and that of the Severn have illustrated the generation of large bed stresses on the inner banks of a bend induced by the rotating flow. These have been related to scouring effects and bank collapse in the case of the Severn. Such conclusion is in accordance with recent findings published by Fukuoka (2000), who has expressed concerns regarding the usual choice of bridge pier location on the inner bank of bends in Japan. Using the calculation from this work, it would appear that the risk of scouring effect in the inner side of the bend is very high during floods, which would jeopardise the survival of any structure constructed at this location in extreme flood conditions. This type of prediction requires the use of a fully three-dimensional code however.

- The present work demonstrates that there is little scale issue in the determination of such flood flow from a fluid mechanics point of view. Indeed the models of the FCF and rivers exhibit similar features to those described above, which implies that the conclusions and recommendations of the FCF programme would be adequate for design purposes for channels of similar proportions.

- It was also confirmed that turbulence transport has a limited impact on the determination of flood flow in meandering channels, which is mostly geometry-driven, Chapter 5. A direct consequence is that simple turbulence models should provide sufficient accuracy for the prediction of flood flow in meandering rivers. As illustrated in Chapter 5 and 6, this is not the case in straight reaches, where weak, turbulence-induced recirculation significantly alters the velocity profiles.

### 7.2.3 General

- This thesis has established that CFD can be used successfully to model detailed open-channel laboratory experiments, Chapter 5, and that it has the potential to either complement or eventually substitute for some physical models. The author in the presentation of results of the Flood Channel Facility (FCF) experiment B23 has made a strong case, for the solution of a flood flow in a meandering two-stage channel with a depth ratio of 25%. Velocity predictions appeared to be within 5% to 10% of the measured values, and the peak shear stress was also accurately predicted using CFX outputs.

- Two models of short river reaches have been constructed to investigate the use of CFD in river engineering for the scenario of a flood flow event resulting in highly three-dimensional flows. In particular the model of the Severn presented in Chapter 6 clearly supports the idea that CFD techniques could be employed in industry to investigate localised reaches of natural channels. The comparison of the model predictions against observations, albeit uncertain, revealed errors of 20% to 30% in the model, although the general flow pattern predicted by both TELEMAC and CFX appeared to be consistent with field observations. Shear stress predictions were also in agreement with physical evidence from the site.

- However, channel complexity and scale impair the implementation of CFD for large-scale problems, especially when using structured grids. Consequently one of the best applications of CFD to river engineering probably remains the investigation of complex flows around man-made structures where perfect resolution and boundary conditions can be implemented.

- Although a quasi three-dimensional approach is more flexible and provided a good general picture of the velocity in two-stage channels, this work has illustrated that only a fully three-dimensional treatment can offer an adequate prediction of the hydrodynamics and the resulting shear stresses on the boundaries.

## 7.3 RECOMMENDATIONS FOR FURTHER RESEARCH

The present thesis has presented a detailed case-study of the potential of CFD for use in flood flow modelling in meandering two-stage channels, some being natural river reaches. In doing so it took CFD further in terms of its traditional application, and, in some aspects, reached some of the limits of the current techniques. This is a natural process when applying techniques borrowed from other disciplines to a new field (Samuels, 1996), and it helped highlight some of the requirements for further research. The author identified six of them:

- More validation work is needed on natural channels, to ensure that some of the conclusions drawn by the author from incomplete data or by analogy with the Flood Channel Facility (FCF) are fully valid. It would be interesting to select one (or a few within a larger research group) site(s) and measure it (them) thoroughly instead of trying to monitor several sites at once with limited resources. However the author is well aware that research usually needs to fit within a given time period (e.g. 3 years) and to be carried out within a limited budget. Nevertheless data collection, at small and large scales, remains an essential step towards numerical code development in river modelling. It is also necessary for the modeller to direct the collection of data, because he knows which data will enable him to test the validity of a CFD model.

- The evaluation of roughness and the improvement of the representation of wall friction at the riverbed also require further research. Being something specific to their field of CFD, it is highly unlikely that civil engineers will find the adequate information in the fields of applied mathematics, aeronautical or mechanical engineering. Experiments should be carried out to understand the role of coarse gravel and the impact of bed form, vegetation and turbulence on the flow. Similarly the impact of the numerical discretization and models on the momentum loss factor should be investigated. Finally, if coarse grids need to be used for practical reasons in river engineering models, new formulations should be researched to implement a better velocity transition at the walls.

- The development of unstructured grids is important to allow a better representation of irregular geometries and the implementation of suitable grids. Mesh adaptation issues should also be investigated to optimise the resources available to model three-dimensional flows. A framework based on multigrid is discussed below.

- Although a rigid lid has been implemented for the representation of the free surface in the present thesis, it does not represent a faithful model of the free surface and is bound to have some impact on the solution, especially where the free surface fluctuates. An improvement of the present method would be to couple the pressure field on the lid to a mesh-adaptation algorithm.

- In this respect it would also be worth investigating recent work in the field of multigrid, which would allow an automatic mesh adaptation based on the boundary conditions and the reduction of the residual in the numerical solution (Trottenberg *et al.*, 2001). It would seem quite natural to let the reduction in the residuals drive the meshing, as multigrid is able to determine where the initial grid needs coarsening or refining to achieve convergence more efficiently.

- Although there exists excellent guides regarding verification and validation in CFD (e.g. AIAA, 1998), the author believes it would be useful for the civil engineering community to produce its own "code of good practice". It would be more suited to our field of application, which usually involves large-scale problems with high uncertainty in the boundary conditions. It would also contribute to address the end-users needs in a more practical way.

## 7.4 REFERENCES FOR CHAPTER 7

1. Abbott, M.B., (1989), "Review of Recent Developments in Coastal Modelling", Proc. Int. Conf. on Hydraulic and Environmental Modelling of Coastal, Estuarine and River Waters, Bradford, England.

2. AIAA (1998) "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations", AIAA Guide G-077-1998, AIAA.
3. Alfrink, B.J., van Rijn, L.C., (1983), "Two-equation Turbulence Model for Flow in Trenches", J. Hydr. Eng., Vol. 109, No. 3, pp. 941-988.
4. Bradbrook, K.F., Biron, P.M., Lane, S.N., Richards, K.S., Roy, A.G., (1998), "Investigation of Controls on Secondary Circulation in a Simple Confluence Geometry Using a Three-Dimensional Numerical Model", Hydr. Processes, Vol. 12, pp. 1371-1396.
5. Easom, G., (2000), "Improved Turbulence Models Wind Engineering" Ph.D. Thesis, The University of Nottingham.
6. Fukuoka, S., (2000), "Design of Compound Meandering Rivers", Third UK/Japan Joint Seminar on Flow and *Sediment Transport in Compound Channels, 10-21 July* 2000, Sapporo, Hokkaido, Japan.
7. Hodskinson, A., Ferguson, R.I., (1998), "Numerical Modelling of Separated Flow in River Bends: Model Testing and Experimental investigation of geometric Controls on the Extent of Flow Separation at the Concave Bank", Hydr. Processes, Vol. 12, pp. 1323-1338.
8. Lane, S.N., Bradbrook, K.F., Richards, K.S., Biron, P.A., Roy, A.G., (1999), "The Application of Computational Fluid dynamics to Natural River Channels: Three-Dimensional versus Two-Dimensional Approaches", Geomorphology, Vol. 29, pp. 1-20.
9. Ma-Lin, (2000), *personal communication.*
10. Meselhe, E.A., Sotiropoulos, F., (2000) "Three-dimensional Numerical Model for open-Channels with Free-Surface Variations", J. Hydr. Res., Vol. 38, No. 2, pp. 115-121.
11. Samuels, P.G., (1996) "Future Modelling Needs – Discussion and Workshop Conclusions", HR Wallingford Paper Reproduced from Proc. First Expert Meeting, Ribamod, River Basin Modelling, Management and Flood Mitigation Concerted Action, 10-11 October 1996, Copenhagen.
12. Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., (1998), "Three-Dimensional Numerical Model for Flow through Natural Rivers", J. Hydr. Eng., Vol. 124, No. 1, pp. 13-24.

13. Trottenberg, U., Oosterlee, C.W., Schüller, A., (2001), "Multigrid", Academic Press.