Guidelines for applying commercial CFD software to open channel flow

July 2005

Guidelines based on the research work conducted under EPSRC Grants GR/R43716/01 and GR/R43723/01: Modelling of open channel flow to illustrate the effects of channel shape and heterogeneous roughness.

Principal Investigators: Professor DW Knight, Dr NG Wright and Dr HP Morvan.
Researchers: Dr X Tang and Dr AJ Crossley.

Published at http://www.nottingham.ac.uk/cfd/ocf/guidelines.pdf
Summary
An investigation has been made into the flow in open channels using a commercial Computational Fluid Dynamics (CFD) package as is commonly used in industrial consultancies. The study has focused on the ability of the software to correctly predict the complex flow phenomena that occur in channel flows. In this project, the predictions have been compared with high quality flume measurements, obtained for a range of simple and compound prismatic channels in order to investigate in detail the ease of mode construction, the incorporation of physical phenomena and numerical accuracy. A comparison has been made between the different turbulence models available. The results show that whilst all the models generally give similar predictions for the bulk features of the flow, there is a marked difference between the secondary flow characteristics, with the accuracy of the predictions increasing with the level of complexity of the turbulence model used. Results from a LES model have confirmed the importance of turbulence modelling.

The results from this study have been used to provide advice and guidelines to practitioners wishing to apply CFD to open channel flows.

1 Introduction
The features inherent in open channel flow result from the complex interaction between the fluid and a number of mechanisms including the channel bed and walls, friction, gravity and turbulence. In the past such flows have been modelled using simplified models (for example the Saint Venant equations), which predict mean characteristics of the flow and often contain a high level of empiricism. These 1D models contain significant simplification which has been investigated experimentally. Of particular importance is the work carried out on the Flood Channel Facility (FCF), details of which are given in Knight and Sellin[1], Knight and Shiono[2] and Knight[3]. These results have been used to develop a number of 1-D methods, such as the Coherence Method (COHM) of Ackers[4-6], the Weighted Divided Channel Method (WDCM) of Lambert and Myers[7] and the Shiono and Knight Method (SKM) elsewhere[8-10]. Many of these methods have been used to derive modifications to the 1D equations. In particular, the SKM has been used to take some account of secondary flow and interactions between the main channel and floodplains. With advances in computer power, interest has risen in applying more sophisticated techniques providing more accurate results and more in-depth information. In other fluid flow fields such as aeronautics, the implementation of more complex models has mirrored the advances in computer technology and 3D models are now commonly used. However this transition has not occurred as rapidly in open channel flow modelling, and most hydraulics models are either 1D or 2D with very few applications of 3D models. This is, in part, due to the inherent difficulties found in
applying CFD in a natural river channel[11]: inter alia, irregular geometry, free surface, vegetation and roughness representation[12].

In this work the application of a commercial Computational Fluid Dynamics (CFD) package to open channel flows has been considered. The software includes various models to solve general fluid flow problems and is widely accepted as a modelling tool in other fields. In this study, the Reynolds Averaged Navier-Stokes equations[13] (RANS) have been used to represent the fully developed flow in a prismatic channel. The RANS equations are obtained by applying time averaging to the full Navier-Stokes equations, which results in six new terms known as the Reynolds stresses. A turbulence model is then needed to account for the Reynolds stresses in order to close the system of equations. In industry the accepted standard is the two equation model $k$-$\varepsilon$ and its derivatives, which assume that the turbulence is isotropic. More complex models exist which account for the anisotropic nature of turbulence, but at an increased computational cost to the user which has often led users to reject this option. However, in the context considered here, the very nature of the secondary flow is driven by the anisotropy of the turbulence and several different turbulence models have been evaluated in addition to the $k$-$\varepsilon$ model such as various standard Reynolds stress models and a new $\omega$-based Reynolds stress model. The technique of Large Eddy Simulation does not rely on Reynolds averaging and directly predicts larger eddies whilst using a model for smaller ones[14].

The research programme was conducted by the Universities of Birmingham and Nottingham and involved detailed CFD experiments for comparison with high quality experimental data collected at Birmingham[15-17]. The latter encompass discharge, velocities and shear stresses. The investigation has focused on prismatic channels which whilst being a simple geometry involve complex fluid flow patterns. Furthermore, 1D representations of conveyance are based primarily on this type of channel. In the case of simple channels, the patterns are similar to those found in a rectangular open channel[18] and generic informative analogies can be drawn.

The CFD investigations have examined the influence of grid size and structure, discretisation of the fluxes, turbulence model, free surface and wall boundaries. The research programme has focused on generating the correct qualitative and quantitative flow features from the CFD software. Bulk quantities such as discharge (bulk velocity), maximum velocity and mean shear stress have been used to ascertain the bulk quality of the predictions, in respect to the experimental data. In addition, consideration has also been given to the distributions of velocity, bed shear stress, Reynolds stresses and vorticity, with particular emphasis on generating the secondary flow patterns that should be present in a prismatic channel.
Results have demonstrated that the CFD is able to correctly predict the mean shear stress and subsequent shear force on the channel bed and banks. However, with standard turbulence models it is not possible to obtain the same shear stress profiles across the channel as found in the experiment. This is because the standard turbulence models are not capable of capturing the details of the phenomena which occur in the types of channel under consideration[2, 15]. In addition, it should be noted that the shear stress calculated from the theory of CFD is conceptually different from that measured experimentally: this is discussed briefly later in this report and in more detail elsewhere[19]. The study has also highlighted the complexities involved in obtaining the correct secondary flow pattern. The currents which are known to be induced by turbulence and the wall effects in open channel flow, should lead to the maximum velocity occurring below the free surface. However in all of the results obtained using the standard treatment of a symmetry plane for the free surface, the maximum velocity has occurred at the free surface. A modified free surface condition is applied here, based on Celik and Rodi [20], which addresses this issue. This condition treats all but the turbulent kinetic energy and turbulent eddy dissipation as symmetry functions; the turbulence entities are computed to account for the lateral distribution of the turbulent kinetic energy at the free surface. Initial results obtained using a Large Eddy Simulation (LES) model at the University of Karlsruhe have demonstrated that LES can predict open channel flow phenomena more successfully than RANS based models. This is not surprising, given that planform vortex formation and shedding is a priori excluded from all Reynolds averaged Navier-Stokes (RANS) equation models, but are known to be present in most overbank flows (e.g. Thomas and Williams[21]; Ikeda et al.[22]; Knight[23]; van Prooijen[24]; van Prooijen et al.[25]). The importance of including vorticity in 3-D turbulence models is highlighted by Tominaga and Knight[26]. One of the main contributions of LES to the present work is the computation of counter-rotating vertical structures in the upward direction in the centre of the channels, leading to the computation of a trough in the wall shear pattern, as observed in the experiments.

2 CFD

There are three key stages involved in CFD modelling. The first is the pre-processing stage, which involves defining the problem through creating the geometry and setting up the grid or mesh, and defining the physics of the problem (including the boundary conditions and the turbulence model). The second stage is the application of the solver whereby the software uses the information provided at the pre-processing stage to generate a numerical solution. Finally the post-processing takes place, where the results are visualised and analysed.
This report is primarily concerned with noting the methods used at the pre-processing stage to define open channel flow problems and also some mention is made of monitoring the solution and verifying that the results are converged. Illustrations are provided to demonstrate how the choice of turbulence model and boundary conditions affect the results.

Although the work detailed in this report was conducted using the CFX software produced by ANSYS[27], the guidelines are intended to be generic and independent of the CFD platform used.

The following sections detail the practices adopted in this project and provide general guidelines for other CFD users in this field.

2.1 Creating the geometry

During the work it was decided to adopt a consistent frame of reference for the coordinate axis such that the $x$ direction corresponded the streamwise direction, the $y$ axis was aligned with the lateral direction and the $z$ axis represented the vertical component. Furthermore the origin was placed at the upstream boundary and coincided with the base of the centreline of the channel, and the water flowed along the positive direction of the $x$ axis. The set up is illustrated in Figure 1 for a simple rectangular channel.

Figure 1. Generic channel geometry.

One particular geometry on which this study focused was Experiment 16 from the PhD work of Kenneth Yuen [16], which is a simple “smooth” trapezoidal channel, for which the cross-section is shown in Figure 2.
Figure 2. Cross-sectional geometry for Experiment 16.

As can be seen from the figure, this channel is 0.15m high, and 0.45m wide at the free surface. A channel length of 0.1m (i.e. similar dimensions to the cross-section) has been used for this geometry. In addition the fact that the channel is symmetric has been exploited such that the flow in half of the geometry is modelled by utilising a symmetry plane along the centre of the channel. A number of tests were conducted to ensure that the results obtained from modelling the full channel were identical to those provided by the half channel simulations. The computational geometry is indicated in Figure 3.

Figure 3. Computational geometry.
A second geometry of a compound channel is also considered. This is shown in Figure 4 and is experiment 10 from the work of Yuen[16, 17]. This was modelled in a similar manner to the simple channel.

![Cross-section geometry for the compound channel.](image)

**Figure 4: Cross-section geometry for the compound channel.**

The approach adopted to build the geometry has been to create the upstream cross-section (noting that the channel is prismatic) and to extend this shape along the channel length. In general, the precise method of construction is dependent upon the software used. Throughout this work, CFX Build was utilised for the model construction.

An additional consideration during the model construction is to identify within the software any entity of the geometry which need to be identified for future reference. This may be because a particular boundary condition needs to be applied to a surface, or because the user wishes to conduct some analysis for a particular domain. In terms of the geometry illustrated in Figure 3, six distinct surfaces can be identified:

- Inlet
- Outlet
- Free surface
- Side wall
- Channel bottom
- Centreline

It is usually judicious to refer to these geometrical entities using spatial locations which are easy to identify. Further reference will be made to these in the next section.
2.2 Creating the mesh

Another issue closely associated with constructing the geometry is setting up the grid. This involves sub-dividing the geometry into cells or elements in order allow for a discrete representation of the domain using points at which the variables will be computed numerically. Throughout this work, hexahedral meshes have been utilised and this has permitted exact specification of the mesh. This has been relatively easy to implement given the nature of the geometries being considered. An alternative strategy is to use tetrahedral elements. This method has the advantage that complex domains can be meshed more easily, but generally speaking the user has less control over the mesh and control and accuracy is reduced. Further, a greater number of cells is usually required. Hence this strategy was not adopted here. An indication of the types of mesh used in this work is shown in Figure 4.

Figure 5. Illustrative mesh cross-section for prismatic channels.

In the case of Experiment 16, mesh construction endeavoured to ensure that the cells at the wall have a uniform size normal to the channel wall, and that the mesh is as uniform as possible overall. This strategy has been implemented by having cells of uniform height (z-direction), uniform width (y-direction) along the base of the channel, and specifying that the cell width of the cells along the free surface follow a
geometric progression whereby the centre cell is three times the width of the cell at the corner. Note that this ratio corresponds to that between the free surface ($B$) and the base ($b$) of the channel. Given that periodic boundary conditions have been imposed, fewer cells are needed in the streamwise direction than would be the case if a developing flow was simulated and so ten elements have been used in the streamwise direction. One point to note is that by using a hexahedral mesh in this fashion, the cells nearest the wall are “skewed”. Ideally the more orthogonal the cells are the better and skewed cells should be avoided. However with the trapezoidal geometries and the objective to maintain fine first cells in the direction off the walls this is difficult to avoid and the skewness is at a reasonable level.

Another consideration is that the mesh should be sufficiently fine to capture the features of the flow. It is difficult to determine if this is the case a priori, and generally a solution is obtained on a coarse mesh and subsequently the mesh is refined and the solutions are compared. The mesh undergoes further refinement until two successive meshes produce solutions that do not differ significantly i.e. there must be a convergence to zero for the normalised difference in key parameter values obtained at the same time and location on two consecutive grids in the series. The process of verifying that the choice of mesh does not influence the solution is known as mesh or grid independence. In addition it is necessary to provide sufficient mesh resolution at the walls, which can be characterised by the $y^+$ values which are a function of the mesh spacing (normal distance from the wall), the flow velocity and viscosity. Generally speaking the aim is to define the mesh such that the $y^+$ values for the cells nearest the wall lie within a certain range. The specific criteria will depend upon the choice of turbulence model and the guidelines provided in the reference manual should be adhered to. In simulations where standard turbulence models with wall functions are used, the standard practice is that the boundary layer in the near wall region should be resolved by a minimum of ten nodes, and that the $y^+$ values should lie in the range $30 \leq y^+ \leq 300$. In the case of turbulence models which integrate the equations through to the wall (such as the SMC-$\omega$ model), a finer resolution is needed due to the computations on extra terms in the model and a minimum of fifteen nodes is suggested by CFX[27] with $y^+$ less than two at the walls. These models therefore allow a proper computation on the boundary layer, leading to enhanced results for the shear, drag and lift induced by the geometry. Much of the CFD results shown here were conducted in a parallel computing environment which allowed for the fine meshes and long run times necessary.

Additional criteria can be specified for characterising how good or bad a mesh is and details of these should be provided within the documentation of any commercial CFD software and are discussed in Morvan [28]. In particular it is important that there not be significant variation between the dimensions of one cell and any of its neighbours.
(the general recommendation is to avoid any ratio greater than 1.2), and that the aspect ratio of any cell is not too different from unity. Within the context of this work, the use of periodic conditions has enabled the use of cells with high aspect ratios because the flow is uniform in the streamwise direction; however, this practice is not to be recommended in general.

2.3 Defining the problem

Having built the geometry and obtained a mesh, the next stage is to define the physics of the problem. This involves such things as specifying the initial and boundary conditions, defining any body forces (e.g. gravity) acting on the flow, the type of fluid concerned and its properties etc.. There is also the issue of specifying the type of model to be used and necessary associated data.

The first consideration is whether the simulation will be steady or transient. This report is primarily concerned with modelling steady state problems, however some consideration has been given to using a Large Eddy Simulation (LES). With the RANS models used in this work, unsteady effects have been averaged so time-dependent solutions were not obtained. In cases with large coherent eddy structures, it is feasible to use unsteady RANS methods, but this is questionable.

The simulations conducted as part of this project have been based on modelling a single phase of water which has generally been taken as at room temperature. More complex options are available such as two phase flows whereby the water and the air above the water are both modelled and the position of the free surface can be tracked. Such simulations are more complex and computationally demanding than modelling a single phase. Furthermore there can be significant inaccuracy in predicting the free surface position. In this project the condition of uniform flow is used to specify the position of the free surface at the depth recorded in the experiments. The use of periodic boundary conditions means that whilst depth is specified, discharge is not and in this way the CFD will calculate a velocity distribution that is consistent with the depth, channel shape and surface roughness. This discharge can then be compared with the experimentally measured value. As a precautionary measure pressure can subsequently be checked on the free surface.

A crucial part of any CFD simulation is specifying what drives the flow. Initial work attempted to reproduce the experimental data by conducting a fully three dimensional simulation in which an average velocity was specified uniformly across the cross-section at the upstream end of the channel and the flow then developed a spatially varying profile across cross-sections further down the channel. It was found that significant channel lengths (leading to increased run times) were needed if this
approach is adopted. This is illustrated in Figures 5 and 6, which show the developing velocity profile for a 200m channel, together with the maximum recorded velocity at different points along the channel when channels of different length are used. Note that in this instance, the total width of the channel is 10m, and the flow depth is 0.25m. The results from Figure 6 suggest that for this case, a longer channel length of 300m is necessary to allow the flow to fully develop and capture the correct maximum velocity. Furthermore this does not tally with the spirit of the experimental set up in which the bed slope was adjusted to match given bulk flow conditions. As a result of this, large meshes had to be generated such that the mesh resolution in the streamwise direction was comparable to that used for the cross-section of the channel. Subsequently it was decided to adopt periodic boundary conditions whereby the solution obtained was did not vary along the length of the channel, and this meant that much shorter channel lengths could be used coupled with fewer cells along the streamwise direction. This meant that more cells could be concentrated on the cross-section, which was the region of interest in the study. As a consequence of employing periodic boundary conditions, it was then necessary to drive the flow via a momentum source (representing the effect of gravity due to the slope of the channel). It is worth noting that the use of a body force representing gravity should not be combined with a specified inlet velocity as this represents an over specification of the model: thus one approach or the other has to be adopted. In the first instance, the slope of the channel was incorporated into the channel geometry, such that the tangent of the angle between the streamwise direction and the base of the channel was equivalent to the channel slope. However, in order to simplify slope changes for different cases, an approach was subsequently adopted whereby the channel was considered to be “flat” and a resolved gravity vector with components both normal and perpendicular to the channel was implemented. Given that the angle $\theta$ represents the angle between the base of the channel and the horizontal such that $\tan \theta$ is equal to the slope of the channel, then given the coordinate frame adopted, the gravity vector is resolved as

$$\rho g = (\rho g \sin \theta, \ 0, \ -\rho g \cos \theta)^T.$$  

Note that the $x$ component causes the water to flow along the channel and that the $z$ component is responsible for creating the hydrostatic pressure. Given that the angle $\theta$ can be quite small, this implies that the $x$ component is relatively small (of the order of 10) whilst the $z$ component can be quite large (of the order of $10^4$). In the simulations conducted it has been found that the $z$ component of the gravity vector can cause convergence problems for the solver. This is due to the fact that most computational algorithms work using a relative pressure and relative density so as to avoid the use of large source terms: in the present case the vertical gravity component leads to a large pressure term which consequently causes errors. In view of this, and following advice from the vendor, this component was set to zero. This has no effect
on the velocity field or shear stress values and only affects the vertical pressure
distribution.

The remaining issue concerned with specifying the problem is the choice of initial and
boundary conditions. The concept of an initial condition in the case of a steady state
problem is just an initial guess to the flow conditions. The solver can be started from a
zero velocity field, but the time in which the solution is obtained can be reduced if a
more intuitive value is specified at the start of the simulation. The option adopted for
this work has been to use the mean velocity from the experiment, and set this to be the
initial condition everywhere throughout the channel.

At the mesh construction stage, it was decided to model only half the channel and
exploit the symmetry in the steady-state flow, and thus a symmetry condition was
applied along the boundary at the centre of the channel. The channel walls (side wall
and bottom) must be considered and treated accordingly. These should be represented
as non-slip walls, and depending upon the geometry should either be set as smooth or
rough walls, whereby a roughness height will need to be specified in the case of the
latter. The final boundary condition is the treatment of the free surface. The simplest
approach to implement is a symmetry plane condition, which will result in the
velocity contours at the free surface appearing normal to the boundary. This is known
as the “fixed lid” approach. It is known however that the velocity contours should not
be normal to the free surface, and that the maximum velocity lies beneath the free
surface: this is known as the “velocity dip” phenomena. Trying to reproduce this
effect using a CFD simulation is actually quite problematic, and as part of the study
CFX has produced a bespoke boundary condition following the format proposed by
Rodi[29] for modifying the turbulence terms at the free surface.

2.4 Selecting the turbulence model and solver criteria
Having constructed the geometry and defined the physics of problem, the next stage is
to select the various turbulence and solver properties. It is assumed here that most
users will be using a RANS based approach. The appropriate choice of turbulence
model will depend upon the problem itself and what characteristics of the flow the
user wishes to reproduce.

In the case of prismatic channels where there are no geometrical variations along the
channel, the two equation k-ε model will fail to predict any evidence of secondary
flow. This will in turn be evident in the shape of the velocity contours, the secondary
flow vectors and the bed shear stress distribution. This is because the k-ε model
assumes that the turbulence is isotropic, whereas turbulence is known to be
anisotropic. In fact it is the anisotropic behaviour of turbulence as it approaches the
walls and free surface, namely the imbalance in the normal Reynolds stresses, that creates the secondary circulation in a straight trapezoidal channel just as it is in a straight duct[18, 30]. In order to see the effects of turbulence in prismatic channels it is necessary to implement a model which solves the Reynolds stresses explicitly. The models falling under this category which have been considered during this project include a Reynolds Stress model proposed by Launder, Rodi and Reece[31] (LRR-IP), a Reynolds Stress model by Speziale, Sarkar and Gatski[32] (SSG) and the Reynolds Stress $\omega$ or SMC-$\omega$ implemented in CFX[27]. In the case of the LRR-IP and SSG models, a similar mesh resolution can be used as for the $k-\varepsilon$ model. Both of these models employ a “law of the wall” at the domain boundaries which connects the flow in the near wall region to the main flow. Employing a wall function avoids having to resolve the near wall region which would require a very fine mesh resolution for the models concerned. Other models, such as those based on an $\omega$ formulation (e.g. SMC-$\omega$) solve the equations throughout the domain and require a finer mesh resolution at the boundaries. Needless to say the choice of turbulence model affects the computational cost of the simulation being performed.

In the case of channels with more complex (non-prismatic) geometry where the secondary flow is more as a result of the shape of the channel than of the turbulence, it is possible to observe the effects of secondary flow without necessarily having to resort to more complex turbulence models. In these circumstances the $k-\varepsilon$ model may produce results comparable to a Reynolds Stress model[33].

The next consideration is the type of numerical scheme to be used to solve the equations. It is recommended to use a higher order scheme as this will give the most accurate results. If difficulties are experienced in obtaining a converged solution, it may then be necessary to resort to using a lower order scheme, but it must be recognised that the results from this are questionable. It is noteworthy that the ASME will not accept papers for publication where the results were obtained with less than second order accuracy[34].

Many solvers work by adopting a pseudo time stepping approach even for steady state calculations; this technique allows for the under-relaxation of the solution during the solution process. This may result in the need for the user to provide a time scale and there are no set rules as to how to calculate this value. In the work conducted by the authors, the general strategy has been to use the automatic time step calculation algorithm which is provided within the software for the initial iterations (of the order of a few hundred in the case of CFX). After a number of iterations this leads to an appropriate reduction of the residual, but it was found that the integrated shear force on the walls continued to fluctuate. Iterations were continued until this parameter became constant and during this stage a user specified value approximately one
quarter to a half of the advection time scale was used. This speeds up the rate at which
a converged solution is obtained. If a user specified time scale value is set and
problems are experienced by the solver, it is generally advisable to see if reducing the
size of the time step improves the situation. The large number of iterations required in
this work is due to the use of periodic boundary conditions in the streamwise
direction.

Other factors which need to be considered before the solution stage can proceed are to
define the convergence criteria and specify any additional quantities which should be
monitored during the simulation. Typically for practical simulations the aim would be
to see the residuals decrease by a factor of $10^4$. Depending upon the properties of the
flow under investigation, it may be necessary to take this limit further to $10^6$ or even
higher particularly if the shear stress distribution is to be considered. As mentioned
above the average shear force on the wall should also be monitored here and used as a
convergence criterion, as in the case of steady uniform flow it is possible to calculate
what the value should be with

$$\tau_0 = \rho g RS_0$$

where $\rho$ is the density of the fluid, $g$ is acceleration due to gravity, $R$ is the hydraulic
radius of the channel and $S_0$ is the slope of the channel. All the results presented have
been converged until the value of the shear stress from the simulation matches this
theoretical value.

During the simulation the user can also observe how various quantities change. In the
case of a steady state problem, as the solution converges specific quantities (i.e.
velocity) at defined locations will tend towards a constant value. Thus the user can
specify a number of monitor points (as they are referred to in CFX) and use these as
additional criteria to monitor the convergence.

### 2.5 Obtaining the solution

Having built the geometry, defined the physics of the problem and selected the
appropriate models, the next stage is to apply the solver in order to obtain the solution.
As noted in the previous section, it is necessary to monitor the residuals as the
solution progresses, and this give an indication of the convergence of the solution.
However it should be noted that additional criteria should also be used, such as
defining monitoring points or checking the average shear stress in the case of steady
uniform flow. In practice it has been observed that the CFD takes some time to obtain
the correct shear stress value and that generally the residuals need to be reduced by a
level beyond that generally recommended within the literature (due to the periodic
boundary conditions). The software documentation should also provide some guidelines on deciding when the solution is converged. Figures 7-9 provide an illustration of the convergence plots during a simulation. Figure 7 shows the combined residual values and Figure 8 shows the monitor points which include the three velocity components at two locations together with the total shear force on the channel boundaries. On first inspection it would appear that the results have converged well and that there was no need to continue the solution beyond 500 iterations. However closer examination of the shear force, as shown in Figure 9, highlights that this property is still changing and that the value does not remain constant until approximately 1300 iterations.

In cases where it proves difficult to obtain a converged solution, the user should try to ascertain what the cause of the problem is before trying to correct the situation. It could be associated with a particular location in which case it may be a feature of the mesh which is causing the problem, in which case the mesh should be modified following the guidance principles of mesh generation. If the user has opted to apply a more sophisticated turbulence model, it is sometimes necessary to obtain an initial solution using a less complex model, and use this as the initial condition. If the user has provided a time scale for the simulation, then this value should be altered to see if that remedies the problem; in the most severe cases the recourse to a systematic CFL condition should be considered for the first few iterations at least. It may also be that the application of certain boundary conditions is at issue, and this needs to be given careful consideration.

Having obtained a converged solution, the user should ensure that the results are mesh independent before placing any emphasis on the results. This can be verified by creating a finer mesh (for example by doubling the number of cells in each direction for a structured grid or by using a systematic mesh adaption process if the software possesses one) and seeing how the solution on that mesh compares to the original. It may also be that certain features in the flow suggest that some form of local refinement in the mesh is necessary.

In practice the procedure for constructing the model and obtaining the results is an iterative one, whereby initial results are obtained on a coarse mesh and subsequent results from a finer mesh are used for the analysis. It is always advisable to start by constructing a simple model and then gradually increasing the complexity whilst testing all each stage.
3 Results

Having obtained a solution, the post-processing software is a powerful tool for visualising the results and conducting the analysis. A variety of flow characteristics can be considered, depending upon what the user wishes to investigate and observe. This particular project has been concerned with the velocity field and the bed shear stress distribution and it has been possible to compare the results obtained with experimental measurements. In general, the user should make an attempt to validate the CFD results with known data so that there can be some confidence in the solution. In the case of open channel flow, the validation is most likely to take the form of a comparison against physical measurements and a qualitative understanding of what features should be present in the flow. As part of the analysis, the user may also wish to perform a sensitivity study and vary any parameters (such as roughness here) which have a degree of uncertainty, and determine what influence they have on the solution.

Post-processing normally focuses on velocities and pressure contours. However, in the present work, the boundary shear stress is of interest. As described above, the iterative solution of the CFD problem is continued until the shear force of the solution matches that of the experiment. However, it is also interesting to study the distribution of shear stress across the channel. This raises the question of how to calculate this quantity from the results. The law of the wall assumes that:

$$\tau_w = \rho \sqrt{C_\mu k}$$

where $C_\mu = 0.09$ (equilibrium) which relates the shear stress to the value of turbulent kinetic energy. It is also possible to use the following formula when the Reynolds stresses are available:

$$\tau_w = \sqrt{\left(\overline{\rho u'v'}\right)^2 + \left(\overline{\rho u'w'}\right)^2}$$

However, in the experimental setup, the shear stress is calculated from the pressure measured with a Preston tube that is calibrated for circular pipe flow. It would not be surprising that these two ways of calculating shear stress give differing answers, particularly in areas where the assumptions behind the law of the wall break down such as at separation points. In the cases considered here, this corresponds to the point on the bank channel side wall where the two secondary circulations meet. For SMC-ω there is a similar assumption in calculating shear stress that makes the shear stress calculation questionable at separation points. This must be borne in mind when the CFD results are compared with experimental measurements.
Figures 10-12 illustrate the results obtained for the trapezoidal channel using three different turbulence models, namely the $k$-$\varepsilon$, SSG and SMC-$\omega$ models. In the results shown, a symmetry plane was used to represent the free surface. As can be seen there is no evidence of re-circulation in the $k$-$\varepsilon$ results. There is some indication of bulging of the velocity contours in the SSG results, whereby the contours tend towards the corner at the bottom of the channel and the interval between the contours is reduced. Such a feature is to be expected and has been observed experimentally[15] and in simulations performed for duct flow[18]. In addition, three re-circulation cells can be identified in the vector plot which indicate the direction for the secondary flow. The bulging is more prominent for the SMC-$\omega$ results and again, three re-circulation cells can be seen. Notice that in all of these results the contours are normal to the free surface which conforms to the boundary conditions used. This is not the case in the results shown in Figures 13-15, where the same models have been used in combination with the modified free surface boundary condition supplied by CFX. In the case of the $k$-$\varepsilon$ model, the contours at the free surface point towards the corner rather than the centre of the channel. The results from the SSG and SMC-$\omega$ models both show the centremost contours pointing inward towards the channel, and it can be verified that the maximum velocity from the SSG model lies beneath the water surface. The use of the modified free surface treatment also effects the location of the re-circulation cells and it can be seen that the bulge in the contour plot for the SMC-$\omega$ results that appears on the side wall is slightly higher than before.

The results obtained from CFX can be compared with the predictions obtained from a LES model – carried out in collaboration with the University of Karlsruhe[35], as shown in Figures 16-18. Note that in the case of a LES model it is necessary to model the entire cross-section. As can be seen, the LES results show a similar bulging of the velocity contours on the side wall as evident in the SMC-$\omega$ results, although the location is higher. There is additional bulging near the channel bed and generally, the magnitude of the non-streamwise velocity components is greater than the results obtained from the RANS based models.

Figures 19-21 show the results from a compound rectangular channel when the modified free surface condition is used. It can be seen that the SSG and SMC-$\omega$ models predict a more significant interaction between the flows on the floodplain and in the main channel. Both models predict the presence of four re-circulation cells where the strongest is located about the intersection between the floodplain and the main channel.

The boundary shear stress predictions for both the simple and compound channels are shown in Figures 22 and 23. Note that in the case of the compound channel, the profile has been plotted as a function of the lateral distance from the centreline ($y + z$)
and that for both geometries, the CFX results have been reflected about the vertical axis to provide values for the full channel. In the case of the trapezoidal channel, the most apparent feature is that the experimental measurements show a decrease in the shear stress at the centre of the channel, whereas the CFX results show an increase. The LES results show very good agreement with the measurements. There is a marked difference between the shape of the profiles obtained from the different turbulence models and for this geometry the SSG results appear to provide the closest match to the data. A feature which is particular apparent in the SMC-ω results, is the local minimum which can be seen on the side wall which corresponds with the location of the bulges in the velocity contours. This phenomenon is not evident in either the experimental data or the LES results. In the case of the compound channel, the SMC-ω predictions appear to give the best correlation to the measurements and the agreement is very good. The most prominent feature in these results is the over prediction by the $k-\varepsilon$ model at the main channel and floodplain intersection. Note that for all of the CFX results, the mean shear stress is equivalent to the value predicted by the theory and measured in the experiments.

Initially results obtained for the various channels assumed the channel boundaries were smooth walls based on the description of the experiments. However, it was observed that the mass flow rates obtained from the simulations were generally higher than the recorded values. This is due to the fact that whilst the experiments were relatively smooth there was still some roughness which needs to be reflected in the CFD. In the results presented, roughness has been applied in the cases of the $k-\varepsilon$ and SSG models in order to obtain the correct mass flow rate and discharge. Typically only a small roughness height (0.5mm) needed to be introduced into the calculations to give the correct value. It has been verified that introducing roughness in this fashion has no impact on the velocity contours or the profile of the boundary shear stress. In the case of the SMC-ω model, it is problematic to apply roughness. It has also been noted that using the modified free surface condition appears to slightly increase the mass flow rate which results in a larger percentage error. Generally the error has been less for the compound channels considered compared to the trapezoidal cases. In the case of trapezoidal channel (with smooth conditions assumed), Experiment 16, the errors using the symmetry plane range from 12-14 %, and from 14-17 % when the modified condition was applied. For the compound channel shown, the range is 2-4% for the modified condition and 1-4% for the symmetry plane. Note however in the case of the compound channel, the SMC-ω model under-predicted the mass flow rate, so using the modified condition improved the result. Generally speaking this model also gave the prediction closest to the experimental value for each of the geometries considered.
4 Conclusions

Results have been shown to demonstrate the performance of a commercial generic CFD package when applied to open channel flows. The results show that the CFD predictions accurately predict the average shear stress value for all of the turbulence models tested. In all of the cases, the mass flow rate was over predicted when smooth walls were applied as expected, with none of the models appearing significantly better than the others.

It was observed that the $k-\varepsilon$ model failed to predict any evidence of secondary flow, and that the standard Reynolds stress approaches indicated some re-circulation with varying degrees, but all were below the expected magnitudes. The most realistic results in terms of the velocity profiles were obtained using a non-standard Reynolds stress model in which the equations were integrated through to the wall without the use of wall functions. A comparison of the bed shear stress distributions showed that whilst all of the models gave the correct average value, none of the models captured the expected profile, which was observed in the experimental measurements. In addition the SMC-$\omega$ appeared to predict features in the profile which were not evident in the experimental and not in line with expectations. In terms of bed shear stress, the SSG model provided the best agreement with the experimental values overall. Whilst the LES results need more analysis the results so far indicate that this approach has considerable benefit compared with the standard RANS models and even the more complex SMC-$\omega$ model. However, it must be borne in mind that this technique requires considerably longer run times, making it impractical for general use at present. It is also noted that the SMC-$\omega$ model is not as well tested as the other models. This along with some of its counter-intuitive results raise questions over its use.

Consideration was also given to utilising a bespoke free surface boundary condition which was developed by CFX, and it was observed that this improved the results compared with the standard symmetry plane condition.

The main conclusion from this study has been that there are limitations as to what can be achieved with CFD using the steady RANS equations coupled with a turbulence model in the case of steady uniform flow. This is because the characteristics of the flow are generated predominantly by the turbulence and its boundary conditions, and the magnitude of the observable quantities is of a similar order to the accuracy with which results can be obtained (a secondary recirculation if of the order of 2-3% of the main velocity). In cases where the channel geometry is more complex and varies along the channel, the secondary flow is more as a result of the geometry than the turbulence and so it is possible to obtain reasonable results using standard turbulence models [30]. Thus the usefulness of CFD and applicability of the models for the
problem under consideration depends very much upon the type of geometry and particularly on the nature of the dominant forces. In the case of 3D flows for irregular channels, such as those considered by Morvan et al.[33], standard RANS based models can successfully predict the characteristics of the flow. In the case of steady uniform flow, such as those illustrated in this report, more sophisticated approaches such as LES are needed in order to predict any quantitative level of detail.

The knowledge and experience gained from this and previous studies has been used to produce a set of guidelines of good practice and checks, are these recommendations are presented in Appendix 1

Acknowledgements

The financial support of the Engineering and Physical Sciences Research Council under grants GR/R43716/01 and GR/R43723/01 is acknowledged. The authors are also grateful to the Ian Jones and Paul Guilbert of ANSYS Europe who assisted with amending the CFX code.

5 References


27. ANSYS-CFX, CFX-5.6 Manual. 2003, Abingdon, Oxfordshire, UK: CFX.
6 Appendix 1

The following should be addressed as good practice in modelling open channel flows with 3D CFD packages:

1. PC requirements
   a. A PC with at least 1Gb of RAM and a fast processor is required. Access to a cluster of PCs is often necessary to accommodate the fine grids required.

2. Inlet and outlet boundary conditions – two options
   a. A measured inlet velocity profile with a total mass flow outlet condition. This requires tests on the influence of channel length which can lead to long domains and consequent long runtimes.
   b. Periodic conditions on velocity and pressure with a imposed gravity force in straight channels

3. Free surface representation
   a. A symmetry plane condition will not reflect experimental results.
   b. A bespoke free surface condition is required for good validation.
   c. When there is noticeable deformation of the free surface, a moving mesh boundary may be needed[36].
   d. When the free surface deformation is large, a volume of fluid approach[37] may be necessary.

4. Symmetry
   a. Use a symmetry plane along the centerline and model half of a symmetric, prismatic channel based on mean values.
   b. Full channel must be used for LES simulations in a prismatic channel and any simulation in a asymmetric or non prismatic channel

5. Channel boundaries
   a. A small roughness value at the wall may be needed for hydraulically smooth walls to obtain the correct discharge.
   b. $k_e$ is a solver parameter and attempting to set it according to physical measurements is not correct[19].

6. Initial conditions
   a. Mean uniform value

7. Turbulence model
   a. Prismatic channel – Reynolds Stress model or LES may be necessary
   b. Non-prismatic channel – k-ε may be sufficient to capture flow characteristics, but tests against a Reynolds stress model are advisable.

8. Establishing convergence
   a. Monitor residuals to obtain a reduction of at least $10^{-4}$.
   b. Monitor the force on walls
c. use monitor points e.g. velocities at various positions

9. Validating results
   a. compare with experimental or site measurements. For non-uniform, non-prismatic channels attention must be paid to the consequences of channel geometry and its discrete representation[38].
Figure 6. An example of the velocity profile when a uniform condition is applied at the inlet.

Figure 7. Maximum velocity at different points along the channel for different channel lengths.
Figure 8. Combined residual plot.

Figure 9. Monitor point residual plot.

Figure 10. Monitor point residual plot – force only
Figure 11. Velocity contours and secondary velocity vectors from the $k$-$\varepsilon$ model using a symmetry plane to represent the free surface.

Figure 12. Velocity contours and secondary velocity vectors from the SSG model using a symmetry plane to represent the free surface.

Figure 13. Velocity contours and secondary velocity vectors from the SMC-$\omega$ model using a symmetry plane to represent the free surface.
Figure 14. Velocity contours and secondary velocity vectors from the $k$-$\varepsilon$ model using the modified boundary condition to represent the free surface.

Figure 15. Velocity contours and secondary velocity vectors from the SSG model using the modified boundary condition to represent the free surface.

Figure 16. Velocity contours and secondary velocity vectors from the SMC-$\omega$ model using the modified boundary condition to represent the free surface.
Figure 17. LES results for the streamwise velocity component for a trapezoidal channel.

Figure 18. LES results for the spanwise velocity component for a trapezoidal channel.

Figure 19. LES results for the vertical velocity component for a trapezoidal channel.
Figure 20. Velocity contours and secondary velocity vectors from the \textit{k-\varepsilon} model using the modified boundary condition to represent the free surface.

Figure 21. Velocity contours and secondary velocity vectors from the SSG model using the modified boundary condition to represent the free surface.

Figure 22. Velocity contours and secondary velocity vectors from the SMC-\omega model using the modified boundary condition to represent the free surface.
Figure 23. Boundary shear stress comparison between the LES prediction and the CFX results for the simple trapezoidal channel.
Figure 24. Boundary shear stress comparison for the compound rectangular channel.