The Hydrodynamics of Grit Particles Moving Round a Bend using CFD Modelling

Dr Titiksh Patel1 and Dr Laurence W. Gill2

1Department of Civil, Structural and Environmental Engineering, Trinity College, University of Dublin, Ireland. Email: patelt@tcd.ie

2Department of Civil, Structural and Environmental Engineering, Trinity College, University of Dublin, Ireland. Email: gilll@tcd.ie Tel: 00 353 18961047

Received: 12 January 2011, Accepted: 12 March 2011

Abstract

CFD modelling has been used to investigate the secondary flow phenomenon and its effect on grit particles carried in curved open channel flow. A CFD model was calibrated against a case study and then validated against data from a physical model with a 30° bend. The flow was calculated by solving the full Reynolds-averaged Navier-Stokes equations and Reynolds Stress turbulence models with a finite-volume method; the air-water interaction at the free surface was simulated using a Volume of Fluid multi-phase model. The hydrodynamics of grit particles (63 to 2000 µm) moving round a bend was then modelled using Discrete Phase Modelling which was further calibrated using experimental data on grit movement from the physical model. Further parametric investigations were carried out using the numerical model on curved open channels for various angles of bend, flow rates, channel widths and radii of curvature to assess their effect on secondary flows.

Keywords: Open channel flow; Hydrodynamics; Secondary flow; Computational fluid dynamics; Fluid dynamics; Sediment transport.

1. INTRODUCTION

Secondary flows are formed when water in an open channel flows round a bend as a result of the varying centrifugal forces acting on fluid molecules moving at different velocities in a vertical direction. These flows act to redistribute the shear stress on the channel bed which has a pronounced effect on erosion, deposition and sediment transport in river channels. Apart from the classical helical flow, a counter rotating outer bank cell is also normally present which is formed by the free surface and the outer bank. This outer bank cell has a protective effect on the outer bank [1] although, to date, few physical studies have been able to observe the outer bank cell probably due to the lack of high resolution experimentation. Two-dimensional [2, 3, 4]; and three-dimensional CFD methods [5, 6] have been used in the field of hydrology in order to improve understanding of turbulent flow behaviour in open channel flows. This research has shown that two dimensional depth-averaged equations are unable always to predict all the features of the channel and therefore a three-dimensional approach may be necessary to gain good predictions of the flow fields. Equally, the lack of rigorous experimental data has also hampered the verification of investigations to date by means of numerical investigation [7]. Ye and McCorquodale [8] and Patel [9] carried out three dimensional numerical investigations on the experiments done by Hicks et al. [10] on a sloped bank curved channel using a $k-\varepsilon$ turbulence model and found that the agreement between both results was generally good. Rameshwaran and Naden [11, 12] concluded that the two equation $k-\varepsilon$ turbulence model [13] was able to reproduce the central region cell successfully but under-predicted its strength, whilst Booij [14] found that the $k-\varepsilon$ turbulence model could reproduce...
the centre-region cell but was unable to predict the outer bank cell. Indeed, it has been stated by some that the standard k-ε turbulence model cannot accurately predict secondary currents because it assumes an isotropic eddy viscosity model. Hence, more complicated turbulence closures such as the Reynolds Stress Model (RSM) are required to improve the predictions [15, 16]. Cokljat and Younis [17] and more recently Kang and Choi [18] performed numerical tests on straight compound channels and rectangular open channels respectively, finding that an accurate simulation of secondary flows resulting from anisotropic turbulence was better simulated using the RSM, especially when applied to meandering channels. Equally, the discretization of non-linear terms occurring in Reynolds Averaged Navier-Stokes equations (RANS) plays a major role in the accurate predictions of turbulent flows - Leschziner and Rodi [19] showing that numerical schemes based on upwind discretization may introduce numerical diffusion if the flow is skewed relative to the numerical mesh. Nicholas [5] and Ma et al. [6] carried out numerical investigations in open channel flows with bends using the commercial CFD software package FLUENT® and found good agreement with the experimental results. More recently a three-dimensional finite volume modelling approach [20] has been calibrated against detailed laboratory physical models on flow and sediment transport on an S-shaped channel which showed that both k-ε and k-ω turbulence models provided reasonable fits with the k-ω turbulence model resulting in the best predictions. The boundary condition at the water surface has also been identified as a critical issue. Most early studies carried out on open channels assumed that the top surface was fixed allowing no movement of water in the simulations. However, the more realistic scenario of the free movement of water surface can be taken into account whereby the air-water interaction has been modeled using the multiphase Volume of Fluid (VOF) model, as developed by Hirt and Nichols [21]. This has been used successfully to model flows over ship hulls [22] and sloshing tanks [23].

In order to model the movement of particles a Discrete Phase Model (DPM) can be used where the concentration of particles in any flow is less than 10% by volume (for example, municipal wastewater, storm water runoff etc). Once the fluid flow field is established by solving the governing fluid flow equations, a particle force balance equation is used to calculate the trajectory of each discrete particle in a Lagrangian frame of reference. For example, Stovin and Saul [24, 25, 26] have used the DPM method to obtain a statistical distribution of sediment destinations in their studies on combined sewer overflows (CSOs). Their studies highlighted the importance of appropriate specification of DPM boundary conditions in the numerical model and also concluded that the results were highly sensitive to the particle injection location (but less sensitive to particle size or density). Studies by Egarr et al. [27] have also investigated the importance of different shape factors used in drag calculations in the DPM model and their effect on particle trajectory calculations. The ability of the DPM to provide reliable outputs for industrial applications has been demonstrated by Faram and Harwood [28] on sediment interceptors and Jayanti and Narayanan [29] on sedimentation tanks.

This study discusses the validation and calibration of a CFD model to investigate the hydrodynamics of grit moving around a bend in a flat-bottomed channel. The flow field was first calibrated using data from detailed experiments carried out on a physical model by Blankaert and Graf [7]. This calibration was then used to model a curved open channel physical model constructed in the laboratory with a bend angle of 30°. The grit particle hydrodynamics experimental data collected was used to calibrate and further validate the CFD model before detailed parametric investigations were carried out using numerical modelling to simulate secondary flows.

2. MATERIALS AND METHODS
2.1 Case Study
The first stage was to see whether it was possible to model secondary flows accurately using the CFD package FLUENT® using the results from previous research on a physical model that had been highly instrumented. The case study chosen was an experiment which had been carried out on a 120° curved open channel experiment by Blankaert and Graf [7] as seen in Figure 1. The test case was selected to imitate the real flow conditions found in river channels which include variable bed topography. The hydraulic parameters for the fluid flow are also shown in Table 1. Three-dimensional velocity measurements were made on a fine grid of 1360 points (spaced every 5 mm at depth of flow, H/22 and every 3 mm transversally, B/133) in the outer half of the cross-section
at 60° from the bend entrance, using an Acoustic Doppler Velocity Profiler at a sampling frequency of 44.6 Hz.

Table 1. Hydraulic parameters for the case study.

| Radius of curvature (m) | Depth of flow (m) | Channel width (m) | Mean velocity (m/s) | Discharge (L/s) | Reynolds number |
|------------------------|------------------|-------------------|---------------------|-----------------|----------------|
| 2.0                    | 0.114            | 0.4               | 0.38                | 17              | 6.73 x 10^4    |

Figure 1. Geometrical layout of the case study channel (after Blanckaert and Graf [7]).

2.2 Physical model - 30° bend
A 30° curved open channel was then constructed in the laboratory out of perspex with the geometrical layout and the cross-sectional details shown in Figure 2. Inlet flow was from a pump from a reservoir through a stilling chamber. Flow was measured by an electromagnetic flow meter on the pressurised pump return pipe.

Table 2. Hydraulic parameters varied (curved open channel).

| Radius of curvature (m) | Depth of flow (m) | Channel width (m) | Mean velocity (m/s) | Discharge (L/s) | Reynolds number |
|------------------------|------------------|-------------------|---------------------|-----------------|----------------|
| 1.56                   | 0.06 - 0.15      | 0.13              | 0.48 - 0.74         | 4 - 15          | 2.9 - 11.1x10^4 |
The sediment hydrodynamics were established by determining the grit particle removal efficiency for different particle sizes under varying flow conditions. The curvature in the channel alignment generated secondary flows (of Prandtl’s first kind) which caused the grit particles carried in the flow to accumulate at the inside of bend. A sump was located at this point into which the particles could settle out and thus be quantified at the end of each trial. Experiments were carried out on three different flow rates: 4 L/s (0.48 m/s), 5 L/s (0.51 m/s) and 6 L/s (0.54 m/s). Once the flow in the channel had reached equilibrium at the fixed flow rate, the grit sample was added gradually and uniformly from the inlet of the channel. The return of any grit by the pump back into the open channel was prevented by a geotextile membrane over the return reservoir. Once the experiment was concluded, the grit was collected from the sump and drying and sieving procedure was carried out to measure the collected PSD which allowed the final assessment of the particle removal efficiency to be made for each trial. Each set of experiments was repeated at least four times to minimize the effect of error and random variations caused by sieve analysis.

Natural sand (grit) was selected as a solid phase due to its particle size distribution and density. The grit sample was sieved into fractions of 0-63, 63-150, 150-300, 300-425, 425-600, 600-1180, 1180-2000 and 2000-4000 \( \mu \)m \[31\]. The specific gravity for each category of particle size used in the experiments was measured using the BS1377 test \[30\] to provide precise information for the CFD model. The measured density for various particles sizes did not vary to a large extent ranging between 2681 kg/m\(^3\) for the 63-150 \( \mu \)m fraction to 2549 kg/m\(^3\) for the 2000-4000 \( \mu \)m diameter fraction of grit particles respectively. The settling velocity for each grit particle size was calculated using Stokes’ law as:

\[
3\pi \nu \rho_s d_p^2 = \frac{\pi}{6} d_p^3 \rho_s g - \frac{\pi}{6} d_p^3 \rho g
\]

(1)

\[
v_s = \frac{d_p^2}{18 \mu} \left( \rho_s - \rho \right) g
\]

(2)

where, \( \mu \) is the dynamic viscosity of the fluid (N-s/m\(^2\)), \( v_s \) is the settling velocity (m/s), \( d_p \) is the (grit) particle diameter (m), \( \rho_s \) is the particle density (kg/m\(^3\)) and \( \rho \) is the density of fluid (kg/m\(^3\)). The calculated settling velocity did change significantly from the fine to coarse particles with increases of up to three orders of magnitude. The value of settling velocity for 63 and 150 \( \mu \)m particles was 0.0036 and 0.02 m/s respectively; the reason why, for example, particles of size <=150 \( \mu \)m prove very difficult to remove during grit separation process in Wastewater Treatment Works \[32, 33\].
3. CFD PROBLEM FORMULATION

3.1 Flow field

Results from the 120° curved open channel case study were compared at two different grid resolutions of the flow domain. Comparisons between the $k-\varepsilon$ and RSM turbulence model predictions were also demonstrated. The computations were performed on an adaptive grid using the general purpose CFD software, FLUENT® 6.2. The governing flow equations for mass and momentum conservations are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$  \hfill (3)

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} \left[ -\bar{\rho} \bar{u}_i \bar{u}_j \right] \hfill (4)$$

Here, $\delta_{ij}$ is the Kronecker delta, and $-\bar{\rho} \bar{u}_i \bar{u}_j$ is the Reynolds stress.

The Reynolds stress equation is given by:

$$\frac{\partial}{\partial t} \left( \rho \frac{u_i u_j}{\delta_{ij}} \right) + \frac{\partial}{\partial x_k} \left( \rho u_i u_j \frac{\partial u_k}{\partial x_k} \right) = -\frac{\partial}{\partial x_k} \left[ \mu \left( \frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i} - \frac{2}{3} \delta_{ik} \frac{\partial u_j}{\partial x_j} \right) \right] + \frac{\partial}{\partial x_k} \left[ -\bar{\rho} \bar{u}_i \bar{u}_j \right] \hfill (5)$$

The $D_{ij}$, $\phi_{ij}$, and $\epsilon_{ij}$ terms need to be modeled in order to close the equations. It should be noted here that the unsteady solver was used only to yield the steady flow results, and not intended to obtain time-accurate solutions. Time derivative terms were discretized using the first order accurate backward implicit scheme. Convection terms were discretized using the third order Monotone upstream-centered schemes for conservation laws (MUSCL) scheme, while diffusion terms were discretized using the second order accurate central differencing scheme. The pressure-velocity coupling was achieved using the SIMPLE algorithm [34]. The VOF method was employed to simulate the air-water interaction at the free surface and is a type of interface-capturing method which relies on the fact that two or more fluids/phases are not interpenetrating and for each additional phase, a new variable-the volume fraction of the phase in the computational cell-is introduced [21]. The mass conservation equation for the $q^{th}$ phase is given by:

$$\frac{\partial \alpha_q}{\partial t} + \nabla \cdot (\alpha_q \rho) = 0$$  \hfill (6)

It should be noted that the volume fraction equation was not solved for the primary phase, but was based on the constraint that in each cell, the volume fraction of all phases must sum to unity, $\left( \sum_{q=1}^{n} \alpha_q = 1 \right)$.

A single momentum equation was solved throughout the domain, and the resulting velocity field was shared among the phases. The momentum equation depends on the volume fractions of all the phases through the fluid properties, which are determined by the presence of the component phases in each control volume, e.g.,

$$\rho = \alpha_p \rho_p + (1-\alpha_p) \rho_q$$  \hfill (7a)
where, subscripts \( p \) (air) and \( q \) (water) denote the primary and secondary phases respectively for open channel flows. In addition, the complexity of implementing boundary conditions on the surface was avoided since the water surface was the interface between the air and the water. For an unsteady fluid flow, the cells filled with air provided the space for the water when the water level rises. Thus, this method allowed free movement of the water at the interface.

\[ \mu = \alpha_q \mu_q + (1 - \alpha_q)\mu_p \]  

(7b)

3.2 Boundary conditions and solution strategy for the CFD model

Numerical grids were constructed with the Gambit preprocessor available in FLUENT®. The flow domain was divided into a number of non-overlapping control volumes or meshes. The analysis of the case study was carried out on two different meshes, whereby the total number of meshes within the whole flow domain was 773053 (\( G_{C1} \)) and 1,144,662 (\( G_{C2} \)) respectively using a T-grid type meshing due to the uneven bottom topography of the 120° curve channel [1]. The spacing of the grids for the 30° curved open channel case was kept uniform in \( x, y \) and \( z \) directions in the whole domain for all the cases analyzed as shown in Table 3. The uniform or plane bottom surface in the 30° curved channel enabled a structured mesh to be created. This comparison of grid density was carried out to determine the sensitivity of the results to the fineness of the grid which was refined to approximately 4 times of the coarsest grid (\( G_{T3} \)). The CFD model files with grid spacing type (\( G_{T1} \)) took a considerable number of days (>4) to finish the simulations under the convergence criteria (residual error after each iteration for the flow parameters – pressure, velocity) of 0.0001. After comparing the experimental data with all the three grid types and confirming that the results obtained were grid independent, subsequent parametric investigations were carried out using the 5 mm grid spacing. Finer spacing in the longitudinal direction also ensured that the curvature within the flow domain was properly approximated and all the flow features were correctly simulated.

Table 3. Grid spacing details for 30° curved open channel.

| Grid Name | Spacing (mm) | Total No. of cells |
|-----------|-------------|--------------------|
| \( G_{T1} \) | 3 \text{ } 3 \text{ } 3 | 2924947 |
| \( G_{T2} \) | 4 \text{ } 4 \text{ } 4 | 1210110 |
| \( G_{T3} \) | 5 \text{ } 5 \text{ } 5 | 631408 |

The boundary conditions for both the flow domains - case study and 30° curved open channel - were as follows:

(i) **Inlet.**

A boundary condition ‘Open Channel’ available in FLUENT [35] was defined at the inlet. Two separate inlets were defined for air and water with the same group ID. Mass flow rates were defined for both phases depending upon the velocity and the inlet area. The depth of the flow was known in advance from the experimental results which helped to define the free surface level before starting the simulations. The flow domain was initialized with the volume fraction of secondary phase (i.e. water) equal to 1 up to the free surface level. This procedure also aided in the faster convergence of the problem.

(ii) **Outlet.**

A Pressure Outlet boundary condition was applied at the outlet. The pressure was kept at atmospheric pressure (i.e. gauge pressure = 0). Two separate outlets were assigned with the same group ID as similar to the Inlet boundary. The volume fraction of the water fraction at the \( outlet \) boundary was assigned as 1 up to the free surface level and 0 from the top of the free surface level to the top surface for the air phase.

(iii) **Top Surface.**

The boundary condition at the interface was avoided by using the VOF model. The top surface was initially kept symmetrical (as a boundary condition) in
which all the normal gradients ($\partial/\partial z=0$) and the normal components were zero. Once the solution stabilized, the top surface boundary condition was changed to ambient pressure conditions to represent the real flow conditions more accurately. The interface between air and water was modeled using VOF model.

(iv) Wall. The bottom and side surfaces were defined as Wall boundary condition which used the standard wall function, which may be expressed as follows:

$$\frac{u' \mu'}{\tau_w / \rho} = \frac{1}{k} \ln \left( E \frac{\rho u' y_p}{\mu} \right) - \Delta B$$

(8)

where, $u' = C_n u^*/k^{1/2}$ and $\Delta B = \frac{1}{k} \ln f_r$

where, $f_r$ is the roughness function that quantifies the shift of the intercept due to roughness effects, $u'_r$ is the mean velocity at point $P$, $k_p$ is the turbulent kinetic energy at point $p$, $\tau_w$ is wall shear stress, $\mu$ is molecular viscosity, $y_p$ is the distance from the wall, $E \approx 9.79$ is empirical constant and $k \approx 0.4$ is Von Karman’s constant. $\Delta B$ depends on the roughness of the material. A fine grid resolution near the wall was avoided by using the wall function which also helped to reduce computational time. The wall roughness value was set during the case study calibration process and then remained constant (at 0.4 mm) for all other simulations.

3.3 Sediment movement

The movement of grit particles within the domain was carried out using a discrete phase model (DPM) approach. The forces acting on each particle [36] for an open channel carrying grit particles are gravity, buoyancy, drag, lift and friction. Other more minor forces also exist called the virtual mass forces which are the force required to accelerate the fluid surrounding the particle, the force in a rotating frame of reference, and the force due to a temperature gradient. These respective forces were ignored in the present study as the density of fluid was not greater than the density of grit particle, there was no rotating reference frame and no temperature gradient in the flow. In addition, the lift force was neglected as it is recommended for only sub-micron particle range [35].

The force balance equates the particle inertia with the above forces acting on the particle, and can be written (for the $x$ direction in Cartesian coordinates) as:

$$\frac{du_p}{dt} = F_d (u - u_p) + \frac{18 \mu C_D \text{Re}_i}{\rho_p a_p^2} + F_x$$

(9)

where, $F_x$ is an additional acceleration (force/unit particle mass) term, $F_d (u - u_p)$ is the drag force per unit particle mass given by:

$$F_d = \frac{18 \mu C_D \text{Re}_i}{\rho_p a_p^2}$$

(10)

and $\text{Re}_i$ is the relative Reynolds number, defined as

$$\text{Re}_i = \frac{\rho_d a_p |u_p - u|}{\mu}$$

(11)

Here, $u$ is the fluid phase velocity, $u_p$ is the particle velocity, $\mu$ is the molecular viscosity of the fluid, $\rho$ is the fluid density, $\rho_p$ is the density of the particle, $C_D$ is the drag coefficient and $d_p$ is the particle diameter. Frictional force was accounted for by the reflection boundary condition. Similarly, force balance equations were solved in $y$ and $z$ directions respectively. Equation 9 incorporates additional forces ($F_x$) such as the virtual mass force which, as discussed, was neglected.
4. RESULTS AND DISCUSSION

4.1 Comparison with Case Study

A comparison between the simulated normalized depth-averaged values for velocity and discharge and measured ones are presented in Figure 3(a, b) for both grid resolutions which also shows the flow behaviour in detail through the investigated outer half section. The value for normalized depth-averaged velocity \( v_{sn} \) and normalized discharge \( Q_{sn} \) was calculated as:

\[
\begin{align*}
v_{sn} &= \frac{1}{h} \int_{-h}^{h} v(x) \, dx, \\
Q_{sn} &= v_m B h / Q
\end{align*}
\]  

where \( v_{sn} \) is depth-averaged velocity, \( B \) is width, \( h \) is local depth of flow, subscript ‘t’ and ‘b’ denotes free surface and bottom of the channel. It can be seen that the flow remained concentrated over the deeper part of the section and the majority of the discharge flowed through the investigated half-section due to the presence of secondary flows.

Figure 3. Comparison between experimental and simulated normalized depth-averaged velocity \( v_{sn} \) and discharge \( Q_{sn} \) for two different grid resolutions (a) \( G_{T1} \), (b) \( G_{T2} \)

However, the depth-averaged values of the downstream velocity remained almost constant throughout the outer half-section and reduced to zero near the side wall due to the presence of the boundary layer. Comparison between both the plots in Figure 3 also confirms that the simulated results are grid independent with minimal difference found between the two different grid resolutions. The computational model under predicts the normalized values for velocity and discharge to some extent which has been attributed to two primary reasons; (i) approximation of the bottom surface in order to create the geometry in the FLUENT®, and (ii) closure of RSM in terms of free surface treatment. However, the main features of the flow were well simulated.

The comparison between the experimental and computational results for velocity vectors and contours is shown in Figure 4. It should be noted that before switching to the RSM turbulence model, the solution was obtained using the \( k-\epsilon \) turbulence model which helped to ensure better convergence and stability of the solution and also enabled a comparison to be made between the velocity fields for both turbulence models in detail. The cross-flow velocity vectors (or secondary flows) simulated by \( k-\epsilon \) model are shown in Figure 4(b) which is calculated as \( \sqrt{v_h^2 + v_z^2} \); where, \( v_h \) is the transverse component and \( v_z \) is the vertical component at the 60° section. It can be seen that the \( k-\epsilon \) turbulence model was unable to simulate the outer bank cell but was able to generate the centre region circulation cell (i.e. the secondary flow). This has been attributed to the fact that the \( k-\epsilon \) turbulence model is based on the assumption of isotropic turbulence. The reproduction of the main circulation cell by the \( k-\epsilon \) model also agreed well with the previous studies made by Rameshwaran and Naden [11, 12] and Ye and McCorquodale [8]. The cross-flow vectors simulated by the RSM turbulence model (Figure 4c) in contrast show that the outer bank cell has been
successfully captured and the main flow features were also very well simulated. Unlike the center-region cell, the outer-bank cell has not always been observed either in previous experiments or computational studies on flows in channel bends. One reason for this is that the measuring experimental grid in most previous investigations has been too coarse and so the accuracy was too low to measure small velocities of the order of ~0.01 m s\(^{-1}\). Furthermore, the outer-bank cell has an intermittent behavior and was thus difficult to visualize experimentally. The CFD results shown here for steady state were attained only after running large number of iterations (~1500-2000) for each set of problem and thus, it was only after time-averaging the measured data over long periods that it became discernible. This outer bank cell is significant in natural river channels as it acts as a protection to the outer bank as it keeps the maximum velocity contours away from the outer bank. The comparison of experimental with computational results also shows that the cross flow velocity vectors were a little under-predicted by the computational model; although all the flow features were reasonably well predicted.

![Diagram of flow vectors](image)

Figure 4. Secondary flow vectors for (a) Experimental, (b) Simulated \(k-\varepsilon\) model and (c) Simulated RSM model (GT1).

The comparison between predicted and measured longitudinal velocity contours is shown in Figure 5 (note all plots in Figures 4 and 5 are for half channel width). The experimental contour plot shows that the maximum value was around 0.55 m/s where the same value is 0.44 m/s for the predicted plots by the computational model. The variation of the longitudinal contours was not symmetric and they were shifted towards the outer bank owing to the presence of secondary flows. The location of the maximum velocity contour is also of interest and was found well below the free surface. The horizontal position of the maximum velocity contour was located at the intersection of both the circulation cells which agreed well with the experimental data. The longitudinal contour value decreased to zero near the wall due to boundary layer formation.

![Diagram of flow vectors](image)
4.2 30° Curved Open Channel

The case study (above) identified the methodology to model secondary flows in curved channels which was then employed for the CFD simulation of the 30° curved channel. A CFD model of the 30° curved open channel (Figure 2) was generated in order to simulate the flow field within the curved channel domain. The flow field for the curved channel was compared using the three different grid resolutions (see Table 3) to evaluate the sensitivity of the results to the grid density. Figure 6 shows the simulated secondary flows and longitudinal velocity contours for the different grid resolutions (GT₁, finest grid, GT₂, and GT₃) at a flow rate of 4 L/s which reveals that the prediction of secondary flow vectors and velocity contours at all three grid resolutions were similar. Although small variations existed between the secondary flow vectors for all the three different grid resolutions, this did not significantly influence the model’s prediction on grit hydrodynamics, as discussed later. The same phenomenon can be seen in the longitudinal plots where the velocity contours are skewed towards the outer bank owing to the presence of secondary flows. The maximum velocity contour varied from approximately 0.5 to 0.6 m/s for all the three different grid resolutions and was present at almost the same distance from the outer wall, indicating that the results were grid independent.

Figure 6. (a-c) Secondary flows and (d-e) longitudinal velocity contours for different grid resolutions.

The free surface measurements were also verified using the VOF model and the results in Figure 7 show that the main features of free surface evaluation within the curved grit channel were very well...
simulated where the water level rises at the outer bend. However, a little discrepancy existed between the computational predictions and the experiments measurements whereby the computational model over predicts the water level by approximately 8% at the 30° section.

![Figure 7. Comparison of free surface (water level) with VOF model (G T1).](image)

4.3 30° Curved Open Channel — sediment movement
Once, the flow field was modelled, the grit particles of known physical properties (diameter and particle density) were injected from the inlet of the channel into the domain. The simulation results of whether the particles were captured by the sump or passed through to the main channel outlet was used to validate the model. The CFD model simulations were carried out at three different flow rates of 4, 5 and 6 L/s. The grit capture from the channel for 4 L/s (average velocity of ~0.48 m/s) was compared at the three different grid resolutions (G T1, G T2 and G T3) as shown in Figure 8, which reconfirms that the results were relatively insensitive to the grid resolutions. This also confirms that the CFD model was able to simulate the main feature of the experimental results, i.e. higher particle removal efficiencies with increase in the particle diameters. The CFD model over-predicts the removal efficiency for particles <900 mm to some extent. The difference between experimental and computational results may be due to experimental error, especially for larger particles ≥300 mm as

![Figure 8. Grit Removal Efficiency, (a) Comparison of computational predictions with experimental results for different grid resolution, (b) Mid-range diameter analysis, (c) Grid independence.](image)
their removal would have been expected to be 100%. However, for smaller particles of size \( \leq 150 \) \( \mu \)m, the difference between model and experimental results becomes more pronounced, although close to the range of the error bars from the physical model experiments. This over prediction by the CFD model remained consistent in all simulations of the experimental curved open channel at the different flow rates.

4.4 Sensitivity Analysis

A sensitivity analysis was carried out on various parameters (particle density, time step, friction considerations, injection location and shape factors) in order understand the effect of each parameter in detail, as discussed below.

4.4.1 Effect of Particle Injection Location

The sensitivity of the DPM to the particle injection location was established by injecting particles at different horizontal segments which were named top, centre and bottom (depending upon their vertical locations) and inside, middle and outside (depending upon their horizontal locations). Apart from the location of the particles, various other authors [24, 25, 26, 28] have also suggested that a large number of particles need to be injected in order to mute the effect of random variations and to obtain representative particle fates. Thus, a minimum of 5000 particles were injected for each set of simulations. In the case of ‘line’ injections, a group of 50 particles were injected 100 times for each horizontal strip and for the ‘point’ injection, a single particle was injected (5000 times) from the bottom of the inlet in order to establish a detailed comparison between different locations.

Figure 9 shows the variation of grit removal efficiency at various locations for different grid resolutions for a flow rate of 4 L/s. Comparatively better grit removal efficiencies were obtained using the ‘line’ injection simulation for particles (especially \( \leq 300 \) \( \mu \)m) which were injected from the bottom of the inlet, compared to the particles injected at the centre and top. An interesting phenomenon was also observed for the simulations using ‘point’ injection whereby particles (of size \( \leq 300 \) \( \mu \)m) injected from the inside of the bend had the lowest grit removal efficiency compared to their corresponding outside and centrally located ones. This was attributed to the fact that these particles were already present at the inside of the bend and so had more probability of getting lifted by the uplift velocity of the secondary flow at the wall (thus avoiding capture by the sump) than the particles injected from other locations.

Particles hitting any particular plane could be recorded with respect to their \( x \), \( y \) and \( z \) co-ordinates in the DPM and so this was used to show their positions at the 30° plane just before the sump (see Figure 10). This clearly shows that the smaller particles injected from the inside, strike the 30° plane at higher vertical distances (i.e. shallower depths), which reduces their corresponding removal efficiencies, compared to particles injected at the centre or outside. The larger particles (\( > 300 \) \( \mu \)m) however, show that their removal efficiency was relatively independent with respect to particle injection location with the exception of the 2000 \( \mu \)m particles which did not all move across to the sump (being spread over a distance of 8 cm from the inside bend) due to their larger mass and thus higher momentum in the direction of flow.

Figure 9: Grit removal from channel efficiency — particle injection location.
Figure 10. Scatter plot for various particle sizes (0.063 — 2.0mm) hitting the 30° plane.

Figure 11 shows the plan view of different particle trajectories for a 30° angle of bend at 4 L/s. Again, it can be observed that the 63 µm particles were not affected by the secondary flow within the curve domain, remaining well spread throughout the cross-section with most of these particles escaping directly to the main outlet of the channel.
Particles of size 150 µm tended to accumulate increasingly at the inside as they pass through the bend with similar trajectory behaviour to the particle sizes of 300, 425 and 600 µm, striking at the inside half of the bend as they reach the sump. For particles of size 1180 and 2000 µm however, some particles remained well spread throughout the cross-section as evident from their trajectories due to their heavier nature and higher momentum perpendicular to the direction of secondary flows, as confirmed earlier in the Figure 10 scatter plots. The trajectories show that there are no particle tracks after the sump for particle sizes of 300-2000 µm as these particles remain within the inside half of the bend and so get captured by the sump. The effects due to different particle locations have been avoided for all subsequent parametric investigations in this CFD study by injecting particles...
uniformly across the inlet cross sectional area so that removal efficiencies obtained for the different particle sizes were only due to variations in the flow field.

4.4.2 Effect of Shape Factor
The efficiency of the particle shape factor at a flow rate of 4 L/s was also compared using different shape factors of 0.25, 0.5, 0.75 and 1.0 where the shape factor, $\phi$, is,

$$\phi = \frac{s}{S}$$  \hspace{1cm} (14)

where, ‘$s$’ is the surface area of a sphere having the same volume as the particle and ‘$S$’ is the actual surface area of the particle. The higher the value of the shape factor (up to unity) the closer the particle is to a spherical shape; a shape factor of 0.25 indicates a flat particle having maximum surface area for a particular volume. The variation of grit removal efficiencies achieved for different shape factors, as shown in Figure 12, reveals increased removal efficiency with increase in shape factor due to the drag laws being obeyed (i.e. less drag on typical spherical particles). The calibration of the CFD model against the physical model results for the 150 µm particle size was optimised when a shape factor of 0.8 was used, a value that was then used for all the subsequent analysis of grit hydrodynamics.

4.4.3 Friction Considerations at the Wall
When a grit particle hits the channel wall, the particle was reflected back based on the coefficient of restitution (COR) between the particle and the Perspex®. Hence, by including the phenomenon of collision of the particle at the wall, this friction force was taken into account by the reflect boundary condition. Four different COR values (0.25, 0.5, 0.75 and 1.0) were investigated which showed very little difference between simulations whereby the grit removal efficiencies remained almost constant for all particle sizes.

4.5 Parametric Investigations
4.5.1 Effect of Flow Rate
Figure 13 (a, b) compares computational predictions with experimental results for flow rates of 5 L/s (~0.51 m/s) and 6 L/s (~0.55 m/s). The CFD model predicts the same general behaviour as found in the physical model experiments with efficiency decreasing with an increase in the flow rate or average flow velocity. However, the computational model again over predicts the efficiency for all particles, in particular for particle sizes 63 and 150 µm.
The CFD predictions of grit removal efficiency at higher flow rates of 8, 10 and 15 L/s (with respective mean velocities of 0.59, 0.65 and 0.74 m/s) are shown in Figure 13(c) which indicates that the grit removal efficiency decreases as the flows increase. The more muted increase in the average flow velocity affects the efficiency of the grit removal device considerably, especially for particles in the size range of 63 to 300 µm.

4.5.2 Effect of Bend Angle
Three different bend angles (15°, 30° and 45°) were investigated at six different flow rates of 4, 5, 6, 8, 10 and 15 L/s, corresponding to average velocities of 0.48, 0.51, 0.54, 0.59, 0.65 and 0.74 m/s respectively. The results (Figure 14) show that the magnitude of secondary flows increased with the increase of the bend angle for any particular flow rate; the rate of increase of magnitude for secondary flow from 15° to 30° being higher than from 30° to 45°. This trend was observed for all ranges of flow rates from 4 to 15 L/s. The graphs also show that the magnitude of the secondary flows increased with the increase in the flow rate (or average flow velocity). The secondary flow velocities ranged between approximately 0.03 to 0.06 m/s with the maximum values found with the 45° bend at a flow rate of 15 L/s. An interesting phenomenon was observed with the cross-flow velocity vectors at 15 L/s with 30° and 45° bend angles whereby an outer bank cell of a relatively smaller magnitude (compared to the main circulation cell) was found at the intersection of the outer bank and the free surface. A similar pattern of two circulation cells consisting of one main and one outer bank cell was also observed by Blanckaert [1]. This outer bank cell is believed to have a protective effect as it is able to divert the maximum longitudinal velocity contour away from the outer bank, hence preventing scour. Such a phenomenon is crucial while analyzing flow behaviour in natural rivers and estuaries.

The secondary flow velocities appear small compared to the mean flow velocity, but have a significant affect on the redistribution of longitudinal velocity contours as seen in Figure 15 for the various ranges of flow rates analyzed. It should be noted that the secondary flow did not increase greater than 10% of the main flow velocity for any of the cases analyzed. The distribution of longitudinal velocity contours along the cross-section for 15°, 30° or 45° was not symmetric and...
was shifted towards the outer bank (right-hand side of graphs) owing to the formation of secondary flows. However, the effect of secondary flow on the redistribution of longitudinal velocity contours on the 15° bend for all the flow rates is not prominent due to their smaller magnitude (~0.03 m/s). Equally, the distribution of longitudinal velocity contours for the 15° bend was similar to any typical straight channel flow.

Figure 14: Cross-flow velocity vectors for different angles of bend at (a) 4 L/s, (b) 6 L/s, (c) 8 L/s, (d) 10 L/s and (e) 15 L/s. [Note, viewed from downstream side – i.e. outer bends on RHS of plots]
Figure 15. Longitudinal velocity contours for different angles of bend at (a) 4 L/s, (b) 6 L/s, (c) 8 L/s, (d) 10 L/s and (e) 15 L/s.

Another feature revealed in all the velocity contour plots are the maximum velocity contours lying well below the free surface, attributed to the presence of secondary flows (as is standard in open channel flow) and directly related to both the angle of bend and the different flow rates. The contours appeared to be increasingly skewed outwards with an increase in the angle of bend (or secondary flows) and this trend was observed for all the ranges of flow rates analyzed.
corresponding maximum values of velocity contours for various flow rates was found to vary from approximately 0.55 to 0.82 m/s for flow rates of 4 and 15 L/s respectively.

4.5.3 Effect of Channel Width
The effect on secondary flows for three different widths of 0.13, 0.19 and 0.25 m was analyzed at the same average flow velocity 4 L/s (mean velocity = 0.48 m/s) on the 30° bend channel. The results in Figure 16 show that the magnitude of the secondary flow increased with the increase in the width of the channel. The rate of increase was significantly higher from 0.13 to 0.19 m than from 0.19 to 0.25 m. It should be noted that the decrease in the depth was due to the maintaining the flow same flow rate between channel width comparisons which was also successfully simulated by the computational model.

![Cross-flow velocity vectors and longitudinal velocity contours](image)

Figure 16. Cross-flow velocity vectors and longitudinal velocity contours, (a) 0.13 m, (b) 0.19 m, (c) 0.25 m

4.5.4 Effect of Radius of Curvature
Three different radii of curvature were also investigated for the 30° bend: 780, 1560 and 2340 mm giving radius of curvature to width ratios (R_c/B) of 6, 12, and 18 respectively for the 130 mm channel width as shown in Figure 17. The results suggested that a decrease in the R_c/B ratio, leads to an increase in the magnitude of secondary flow. The increase approximately quadrupled from 0.01 m/s for R_c/B = 18 to 0.04 m/s for R_c/B = 6 indicating that the radius of curvature is also a significant factor in the development of secondary flows and consequently in river morphology. However, the magnitude of secondary flow does not seem to change significantly from R_c/B = 6 to 12 compared with R_c/B = 12 to 18 where the magnitude of the secondary flow reduced considerably for R_c/B = 18. The same behaviour can also be seen in the axial velocity contour plots (Figure 17b) where the pattern of velocity contours was quite similar for R_c/B = 6 and 12.
Overall, the successful calibration of the CFD model of the curved channel from physical experiments on both the flow field and hydrodynamics of grit has enabled further geometrical and dynamical parameters (bend angle, radius of curvature, flow rate and channel width) to be investigated with respect to their influence on secondary flows development. This study has indicated that CFD can be a very useful tool in studying complex three-dimensional flow behaviour within both engineered channels and natural rivers and streams. However, it must be appreciated that natural channels normally have non-flat bed profiles and river flows are further complicated by a range of hydrologic, geophysical, geotechnical, geo and biochemical conditions. For example, the flow conditions are unsteady and depth varying, particularly in times of flood conditions.

5. CONCLUSIONS

The comparison between the highly instrumented physically modelled case study and the computational results have shown that the main secondary flow features can be well simulated using CFD techniques. In addition to the main circulation cell, an outer bank cell was also simulated using the RSM turbulence model at the intersection of the outer bank and the free surface. These CFD modelling techniques were then used with a discrete phase model (DPM) approach to simulate the physical model experimental results of grit particles moving round a 30° curved channel. The CFD model was able to simulate the main feature of the experimental results, i.e. higher particle removal efficiencies with increase in the particle diameters, although did slightly over-predict the removal efficiency for smaller grit particles. The model also indicated that, on average, the magnitude of secondary flows varied approximately from 0.03 to 0.06 m/s with the increase in the average channel flow velocity from 0.48 to 0.74 m/s (flow rates of 4 to 15 L/s).

The calibrated CFD model was then used to investigate the effect of different physical parameters on secondary current formation. A decrease in the radius of curvature promoted an increase in the magnitude of the secondary flow. This increase quadrupled from approximately 0.01 m/s for $R_c/B=18$ to 0.04 m/s for $R_c/B=6$. The magnitude of the secondary flow also tended to
increase with an increase in the bend angle, although the increase was more pronounced between bend angles of 15° to 30° than between 30° to 45° - a trend that was observed to be almost consistent for all ranges of flow rates from 4 to 15 L/s. Due to the presence of the secondary flows, the distribution of longitudinal velocity contours was not symmetric and tended to become increasingly skewed towards the outer bank with an increase in the magnitude of secondary flows. The distribution of longitudinal velocity contours for the 15° bend however, was similar to any typical straight channel flow. In addition to the main circulation cell, an outer bank cell was also found in the curved channel at a relatively high depth/curvature (H/Rc) ratio of 0.096 at the flow rate of 15 L/s. Hence, further parametric studies would be helpful in order to understand the formation of the outer bank cell which is crucial with respect to the overall understanding of river flow, meander progression and fluvial erosion processes. In addition, these results indicate that such a channel morphology with a sump at the inside of the bend could have an effective industrial application in the water industry in the form of a sediment interceptor as part of stormwater or wastewater treatment processes.

REFERENCES

[1] K. Blanckaert, Flow and turbulence in sharp open channel bends, PhD Thesis, EPFL, Lausanne, France, 2003.
[2] J.C. Bathurst, C.R. Thorne and R.D. Hey, “Secondary flow and shear stress at bends.” Journal of the Hydraulics Division ASCE, vol. 105, pp. 1277–1295, 1979.
[3] P. Rameshwaran and K. Shiono, “Predictions of velocity and boundary shear stress in compound meandering channels.” In Proceedings of River flow 2002, International Conference on Fluvial Hydraulics, Louvain-la-Neuve, Belgium, pp. 223-231, 2002.
[4] P. Rameshwaran and K. Shiono, “Computer modelling of two-stage meandering channel flows,” Proceedings of the Institution of Civil Engineering, Water and Maritime Engineering, vol. 156, no. 4, 773-788, 2003.
[5] A.P. Nicholas, “Computational fluid dynamics modeling of boundary roughness in gravel-bed rivers: an investigation of the effects of random variability in bed elevation,” Earth Surface Processes and Landforms, vol. 26, pp. 346-362, 2001.
[6] L. Ma, P.J. Ashworth, J.L. Best, L. Elliot, D.B Ingham and L.J. Whitcombe, “Computational fluid dynamics and the physical modeling of an upland urban river,” Geomorphology, vol. 44, pp. 375-391, 2002.
[7] K. Blanckaert and W.H. Graf, “Mean flow and turbulence in open-channel bend,” Journal of Hydraulic Engineering, vol. 127, no. 10, pp. 835-847, 2001.
[8] J. Ye and I.A. McCorquodale, “Simulation of curved open channel flows by 3D hydrodynamic model.” Journal of Hydraulic Engineering, vol. 124, no. 7, pp.687-698, 1998.
[9] T. Patel, A study on hydrodynamics of grit particles in curved open channels using physical and computational modelling, PhD Thesis. Trinity College Dublin, Ireland, 2007.
[10] F.E. Hicks, Y.C. Jin and P.M. Steffler, “Flow near sloped bank in curved channel.” Journal of Hydraulic Engineering, vol. 116, no. 1, pp. 55-70, 1990.
[11] P. Rameshwaran and P.S. Naden, “Modeling turbulent flow in two-stage meandering channels,” Proceedings of the Institution of Civil Engineering, Water Management, vol. 157, pp. 159-173, 2004.
[12] P. Rameshwaran and P.S. Naden, “Three-dimensional modeling of free surface variation in a meandering channel,” Journal of Hydraulic Research, vol. 42, no. 6, pp. 603-615, 2004.
[13] B.E. Launder and D.B. Spalding, “The numerical computation of turbulent flows,” Computer Methods in Applied Mechanics and Engineering, vol. 3, pp. 269-289, 1974.
[14] R. Booj, “Measurements and large eddy simulations of the flows in some curved flumes,” Journal of Turbulence, vol. 4, pp. 1-17, 2003.
[15] B.E. Launder, G.J. Reece and W. Rodi, “Progress in the development of Reynolds stress turbulence closure,” Journal of Fluid Mechanics, vol. 68, no., pp. 537-566, 1975.
[16] I. Nezu, “Open-channel flow turbulence and its research prospect in the 21st century,” Journal of Hydraulic Engineering, vol. 4, pp. 229-246, 2005.
[17] D. Cokljat and A. Younis, “Second-order closure study of open-channel flows,” Journal of Hydraulic Engineering, vol. 121, no. 2, pp. 94-107, 1995.
[18] H. Kang and S-U Choi, “Reynolds stress modelling of rectangular open-channel flow,” International Journal for Numerical Methods in Fluids, vol. 51, pp. 1319-1334, 2006.
M.A. Leschziner and W. Rodi, “Calculation of annular and twin parallel jets using various discretization schemes and turbulence-model variations,” *Journal of Fluid Engineering ASME Transactions*, vol. 103, no. 6, pp. 352-360, 1981.

A. Khosronejad, C. Rennie, S.A.A. Salehi Neyshabouri and R.A. Townsend, “3D numerical modeling of flow and sediment in laboratory channel bends,” *Journal of Hydraulic Engineering*, vol. 133, no. 10, pp. 1123-1134, 2007.

C.W. Hirt and B.D. Nichols, “Volume of Fluid (VOF) method for the dynamics of free boundaries,” *Journal of Computational Physics*, vol. 39, pp. 201-225, 1981.

I. Senocak and G. Iaccarino, “Progress towards RANS simulation of free-surface flow around modern ships,” *Centre of Turbulent Research, Annual Research Briefs*, pp. 151-156, 2005.

S.H. Rhee, “Unstructured grid based Reynolds-Averaged Navier-Stokes method for liquid tank sloshing,” *Transactions of the American Society of Mechanical Engineers*, vol. 127, pp. 572-582, 2005.

R.S. Stovin and A.J. Saul, “Efficiency prediction for storage chambers using computational fluid dynamics,” *Water Science and Technology*, vol. 33, no. 9, pp. 163-170, 1996.

R.S. Stovin and A.J. Saul, “A computational fluid dynamics (CFD) particle technique approach to efficiency prediction.” *Water Science and Technology*, vol. 37, no.1, pp. 285-293, 1998.

R.S. Stovin and A.J. Saul, “Computational fluid dynamics and design of sewage storage chambers,” *Journal of the Chartered Institution of Water and Environmental Management*, vol. 14(Apr.), pp. 103-110, 2000.

D.A. Egarr, M.G. Faram, T. O’Doherty and N. Syred, “An investigation into the factors that determine the efficiency of a hydrodynamic vortex separator,” *Proceedings of NOVATECH Conference*, Lyon, France, pp.1-8, 2004.

M.G. Faram and R. Harwood, “A method for the numerical assessment of sediment interceptors,” *Water Science and Technology*, vol. 47, no. 4, pp. 167-174, 2003.

S. Jayanti and S. Narayanan, “Computational study of particle-eddy interaction in sedimentation tanks,” *Journal of Environmental Engineering*, vol. 130, no. 1, pp. 37-49, 2004.

BS1377, *Methods of test for soils for civil engineering purposes. General requirements and sample preparation*, British Standards Institution, 1990.

M. Rupke, “Hydrodynamic grit separation in search of improved grit capture,” *Proceedings of the Water Environment Association of Ontario (WEAO) Conference*, Windsor, ON, Canada, 1994.

Metcalf and Eddy, Inc., *Wastewater Engineering – Treatment, Disposal & Reuse. 4th Ed*. Tchobanoglous, G., Burton F. L., and Stensel, H.D., McGraw-Hill, Inc., 2003.

G. v. R. Marais and A.C. van Haandel, “Design of grit channels controlled by parshall flumes,”*Water Science and Technology*, vol. 33, no. 3, pp. 195-210, 1996.

S.V. Patankar, *Numerical heat transfer and fluid flow*, Hemisphere Publishing Corporation, Taylor & Francis Group, New York, 1980.

Fluent, *Fluent user manual Version 6.2*, ANSYS Inc., 2006.

H. Chanson, *The hydraulics of open channel flow: an introduction*, Arnold Publishers, London, U.K., 1999.

**Journal of Computational Multiphase Flows**