Analysis of the role of turbulence in curved open-channel flow at different water depths by means of experiments, LES and RANS

W. van Balen a, K. Blanckaert b, W. S. J. Uijttewaal a

a Faculty of Civil Engineering and Geosciences, Delft University of Technology, Delft, The Netherlands
b ICARE-ENAC, Ecole Polytechnique Fédrale, Lausanne, Switzerland

First published on: 23 April 2010


To link to this Article: DOI: 10.1080/14685241003789404
URL: http://dx.doi.org/10.1080/14685241003789404
Analysis of the role of turbulence in curved open-channel flow at different water depths by means of experiments, LES and RANS

W. van Balen\textsuperscript{a,}\textsuperscript{*}, K. Blanckaert\textsuperscript{a,b} and W.S.J. Uijttewaal\textsuperscript{a}

\textsuperscript{a}Faculty of Civil Engineering and Geosciences, Delft University of Technology, Delft, The Netherlands; \textsuperscript{b}ICARE-ENAC, Ecole Polytechnique Fédérale, Lausanne, Switzerland

(Received 19 March 2009; final version received 17 March 2010)

In order to unravel the main flow and secondary flow characteristics and the role of turbulence in a curved single-bend open-channel flow, large-eddy simulations (LES) and Reynolds-averaged numerical simulations (RANS) were carried out and compared with experiments of the flow through a strongly bent laboratory flume. Turbulence was found to play an important role with respect to processes that are important in natural rivers. The strength of the curvature-induced secondary flow in the core of the flow domain, which is the most typical feature of curved open-channel flow, depends on the turbulence. Turbulence is especially important in the flow regions near the banks. Only the LES model is able to resolve accurately the boundary layer detachment and the formation of an internal shear layer at the inner bank as well as the outer-bank cell of secondary flow, whereas the RANS model is unable to reproduce these processes. Turbulence also conditions the magnitude of the bed shear stress, as indicated by the considerable overestimations of the bed shear stress by the RANS model as compared with the LES model. As a result, only LES properly reproduces the experimentally measured overall friction losses over the bend, whereas RANS overestimates these losses. The large scales of turbulence play an essential role in generating these flow processes by means of their interaction with the mean secondary flow structures, whereas small-scale turbulence is merely dissipative and does not play a dynamic role with respect to the time-averaged flow pattern. This implies that the simulated flow field is rather insensitive to the sub-grid model in the LES computation.

Keywords: large-eddy simulation (LES); Reynolds-averaged numerical simulation (RANS); river flow; secondary flow; open-channel flow; curvature; flow separation; outer-bank cell.

1. Introduction

The curvature of streamlines of the flow along a convex or a concave wall has amply been studied in the past because of its large importance for applications in an industrial and a geophysical context. In this respect, many efforts have been put in the study of curved flow through closed ducts to gain insight in the influence of streamline curvature on turbulent flows, which have resulted in comprehensive reviews by, for instance, Bradshaw [1] and Patel and Sotiropoulos [2]. For flow through closed ducts, it is known that instabilities at the concave wall could give rise to Taylor–Görtler vortices (cf. Saric [3]), whereas convex surface curvature could give rise to flow separation (cf. Simpson [4]).

\textsuperscript{*}Corresponding author. Email: wimvanbalentu@gmail.com

ISSN: 1468-5248 online only
© 2010 Taylor & Francis
DOI: 10.1080/14685241003789404
http://www.informaworld.com
A special case of streamline curvature along curved walls is curved open-channel flow. A typical example of this kind of flow is natural river flow. Additional complexities of river flows are usually the complex bed topography and the high Reynolds number. In addition, the flow can often be considered as shallow since the horizontal length scales are much larger than the vertical length scale. Because the shape of the mobile sediment bed is strongly dependent on the flow features, the flow and turbulence properties have to be well understood in order to be able to adequately manage the erosion and sedimentation patterns of rivers as well as to properly predict the spreading of scalar quantities, such as heat or pollution, in river flows.

With respect to the large-eddy simulation (LES) of curved turbulent flows, the amount of work that has been carried out is mainly focused on flow through closed ducts, for instance the work of Moser and Moin [5], Lund and Moin [6], Silva Lopes et al. [7] and Münch and Métais [8]. However, little has been published on the LES of curved open-channel flow, which is typically much shallower (i.e. higher width-to-depth ratio) than flow through curved ducts. To the knowledge of the authors, only the work of Booij [9], Stoesser et al. [10] and Van Balen et al. [11] could be mentioned within this context. Currently available studies on curved open-channel flow are often restricted to Reynolds-averaged numerical simulations (RANS) (cf. [12–14] and many others) with sometimes unsatisfactory results regarding the secondary flow pattern (cf. [9]).

The main aim of the present paper is to contribute to a better understanding of natural river flow by means of a combined approach of experimental work and numerical work by considering the open-channel flow through a strongly curved laboratory flume. For this purpose, the laboratory data of Blanckaert [15] are used. Van Balen et al. [11] have considered mildly curved single-bend open-channel flow, and Stoesser et al. [10] have investigated meandering open-channel flow; the present study broadens the parameter space towards strongly curved single-bend open-channel flows. Although discrepancies are present between natural river flow and flow through a laboratory flume – the flume consists of a rectangular cross sections and the flow through it has a relatively low Reynolds number – the present study is worthwhile since it enables investigating hydrodynamic processes that are important in natural rivers under optimised and controlled conditions.

More specifically, the goal of the present study is to answer the following five questions. Firstly, what is the role of turbulence regarding flow processes in curved open-channel flow? Secondly, what is the performance of LES- and RANS-type computations regarding the simulation of curved open-channel flows? Thirdly, what is the influence of the sub-grid closure model on the outcome of the LES in this respect? Fourthly, what is the structure of the turbulence in curved open-channel flows? And fifthly, what is the influence of the water depth on the flow characteristics in an open-channel bend?

Once the numerical model has been validated, it provides a data set complementary to the experimental results. Combining both the experimental approach and the numerical approach, a more comprehensive and complete view on the physics of curved open-channel flows can be accomplished which can largely contribute to a better understanding of natural river flow, its turbulence structure and its modelling.

The outline of the present paper is as follows. In Section 2, the physical domain, the experimental technique and the numerical model are described as well as other modelling related affairs such as the turbulence closure model and the boundary conditions. The comparison of the primary flow statistics, such as the water levels and the velocity distribution, is elaborated in Section 3. The bed shear stresses are shown in Section 4. Section 5 addresses the structure of the secondary flow along the flume in general and in the centre region,
Table 1. Hydraulic conditions for the shallow case, the medium case and the deep case. \( H \) denotes the water depth, \( R \) the radius at the centre line, \( B \) the width of the flume, \( R_h \) the hydraulic radius, \( Q \) the discharge, \( V_{av} \) the bulk velocity, \( Re \) the Reynolds number and \( Fr \) the Froude number. \( Re \) and \( Fr \) are based on the bulk velocity \( V_{av} \) and the water depth \( H \). Notice that the bulk velocities for the shallow case and the deep case are slightly different in the experiment: 0.40 m/s for the shallow case (\( Q = 56 \) l/s) and 0.38 m/s for the deep case (\( Q = 102 \) l/s).

<table>
<thead>
<tr>
<th>( H ) (m)</th>
<th>( R ) (m)</th>
<th>( B ) (m)</th>
<th>( R_h ) (m)</th>
<th>( Q ) (l/s)</th>
<th>( V_{av} ) (m/s)</th>
<th>( Re )</th>
<th>( Fr )</th>
<th>( R/B )</th>
<th>( R/H )</th>
<th>( B/H )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.108</td>
<td>1.7</td>
<td>1.3</td>
<td>0.093</td>
<td>60</td>
<td>0.43</td>
<td>46440</td>
<td>0.42</td>
<td>1.3</td>
<td>15.7</td>
<td>12.0</td>
</tr>
<tr>
<td>0.159</td>
<td>1.7</td>
<td>1.3</td>
<td>0.128</td>
<td>89</td>
<td>0.43</td>
<td>68400</td>
<td>0.34</td>
<td>1.3</td>
<td>10.7</td>
<td>8.2</td>
</tr>
<tr>
<td>0.206</td>
<td>1.7</td>
<td>1.3</td>
<td>0.156</td>
<td>115</td>
<td>0.43</td>
<td>88580</td>
<td>0.30</td>
<td>1.3</td>
<td>8.3</td>
<td>6.3</td>
</tr>
</tbody>
</table>

the outer-bank region and the inner-bank region in particular. Results for the turbulence structure of the flow are discussed in Section 6.

2. Problem definition

2.1. Physical domain

The geometry of the experimental setup used by Blanckaert [15] is shown in Figure 1. The setup consists of a straight inflow section of 9 m, a curved section of 193° and a straight outflow section of 5 m. The radius of curvature at the centre line of the channel is \( R = 1.7 \) m. The width of the flume is \( B = 1.3 \) m. Experiments have been carried out for three different flow depths of 0.108, 0.159 and 0.206 m respectively, and a similar bulk velocity of \( V_{av} \approx 0.4 \) m/s. These three cases will henceforth be denoted as the shallow case, the medium case and the deep case. The experimental conditions are summarised in Table 1.

The vertical sidewalls are hydraulically smooth and made of polyvinyl chloride, whereas the horizontal bottom is hydraulically rough: it consists of sand particles with a mean diameter of 2 mm spread out and fixed on the bottom by means of spray. With the Nikuradse equivalent roughness \( k_s = 6 \) mm being three times the sand diameter if \( Re \gg 4000 \) according to Van Rijn [16], the flow in all the three experiments is fully rough turbulent.
with

\[ Re_* = \frac{v_* k_*}{\nu} = 260 > 70 \]

and sub-critical as \( Fr < 1 \).

The experiments were conducted with a horizontal bed in order to isolate the influence of the relative curvature parameter \( H/R \) and avoid contamination by the interaction of the flow with a mobile bed topography. The fact that the single-bend geometry is characterised by discontinuities in radius of curvature at the bend entry and exit allows investigating the adaptation of the flow to changes in curvature as well as the recovery of the flow when the curvature vanishes in an isolated way.

2.2. Experimental technique

Measurements of the velocity vector have been carried out with high spatial and temporal resolution by means of an acoustic Doppler velocity profiler (ADVP). The ADVP consists of a central emitter, surrounded by four receivers, placed in a water filled box that touches the water surface by means of an acoustically transparent mylar film. The central transducer periodically emits acoustic pulses. While progressing along the water column, the acoustic pulse is backscattered on targets moving with the water. The backscattered Doppler-shifted echo is recorded by the four receivers to allow for distinguishing the three velocity components. By continuously recording the backscattered signal, the entire water column is covered. Measurements were made with a sampling frequency of 31.25 Hz and a sampling period of 180 s. A more complete description of the working of the ADVP is given by Hurther and Lemmin [17]. Data treatment procedures are described by Blanckaert [18].

Many experimental data are available for the medium case \((H = 0.159 \text{ m})\): measurements were performed at 13 different cross sections in the flume, of which seven cross sections are situated in the bend at 15°, 30°, 60°, 90°, 120°, 150° and 180°. For the two other cases, the experimental data are more scarce: for the shallow case \((H = 0.108 \text{ m})\) only the 135° cross section and for the deep case \((H = 0.206 \text{ m})\) only the 75° cross section were measured. For the shallow case and the deep case, some experimental data are also available at the angular locations 15°, 30°, 60°, 90°, 120°, 150° and 180° in the bend be it only at the centre line. These measurements were made with a sampling period of 90 s.

The relative uncertainty of the measurements is estimated (see [18]) as 4% for the streamwise velocities, 10% for the cross-sectional velocities, 30% for the friction velocity, 15% for the turbulent normal stresses and turbulent kinetic energy and 20% for the turbulent kinetic energy. The accuracy in the ADVP measurements is, moreover, reduced near the flow boundaries. At the water surface, the ADVP device perturbs the flow in a region of about 2 cm (about 10%–20% of the flow depth in the experiments). In a flow layer of about 2 cm near solid boundaries, the ADVP seems to underestimate turbulent characteristics, which is tentatively attributed to the high-velocity gradients within the measuring volume and/or to parasitical echos from the solid boundary. ADVP measurements seem to underestimate systematically the vertical velocity fluctuations.

2.3. Numerical model

For the simulation of curved open-channel flow, we adopt a cylindrical reference system with \( r, \theta \) and \( z \) the transverse, streamwise and vertical direction respectively [see Figure 2(a)].
The corresponding velocity components are the transverse velocity $u$, the stream-wise velocity $v$ and the vertical velocity $w$. In the numerical model, the incompressible Navier–Stokes equations formulated in cylindrical coordinates are solved. In straight parts of the geometry, we let $1/R \to 0$, yielding vanishing centripetal terms, and $r \partial \theta \to \partial y$ for axial partial derivatives. In this way, the coordinate systems in straight parts changes to a Cartesian one with coordinates $x$, $y$ and $z$ being the respective transverse, streamwise and vertical directions [see Figure 2(b)].

The incompressible Navier–Stokes equations are solved on a staggered mesh using the finite-volume paradigm with the pressure-correction algorithm. The equations are numerically integrated using the midpoint rule. Basically, this procedure results in the spatial discretisation of the domain following the second-order central scheme. The equations are integrated in time using the explicit second-order Adams–Bashforth scheme. Details on the numerics can be found in [19] and [20].

### 2.4. Turbulence modelling

#### 2.4.1. The LES approach

As in LES, the proper reproduction of the flow motion on the supra-grid scale requires a model for the closure of the flow motion on the sub-grid scale, either the static Smagorinsky model or the dynamic Smagorinsky model are used in this study. The sub-grid scale viscosity $\nu_{sgs}$, needed for the modelling of the sub-grid scale stress tensor, is modelled using Smagorinsky’s model following:

$$\nu_{sgs} = C_s^2 \Delta^2 \left| \tilde{S}_{ij} \right|,$$

where $C_s$ is Smagorinsky’s constant and $\Delta$ is the turbulence resolution length scale (cf. [21]), defined as $\Delta = (r \Delta r \Delta \theta \Delta z)^{1/3}$ in curved parts of the geometry and $\Delta = (\Delta x \Delta y \Delta z)^{1/3}$ in straight parts, and $\tilde{S}_{ij}$ is the rate of strain tensor based on the resolved velocities. The tilde $\tilde{\cdot}$ denotes that quantities resolved on the supra-grid scale are concerned. In this paper,
several values for Smagorinsky’s constant are used to investigate the sensitivity to the model outcome. A standard Van Driest damping function is used in order to prescribe $\nu_{sgs} \rightarrow 0$ at solid walls.

An alternative to this standard Smagorinsky model is the dynamic Smagorinsky model, as proposed by Germano et al. [22] in the modified version of Lilly [23]. This sub-grid scale model replaces the constant $C_s^2$ in Equation (2) by $C_d$, such that

$$v_{sgs} = C_d \Delta^2 |\tilde{S}_{ij}|.$$  (3)

The model constant $C_d$ is calculated via

$$C_d = \frac{L_{ij} M_{ij}}{M_{ij} M_{ij}},$$  (4)

where $L_{ij}$ and $M_{ij}$ are defined as

$$L_{ij} = \hat{u}_i \hat{u}_j - \tilde{u}_i \tilde{u}_j$$ and $$M_{ij} = 2 \Delta^2 |\tilde{S}| \tilde{S}_{ij} - 2 \Delta^2 |\tilde{S}| \tilde{S}_{ij}.$$  (5)

In these equations, the hat $\hat{\cdot}$ denotes spatial filtering over the test filter, which is defined as the spatial average of the values at a certain cell and its $3^3 - 1$ neighbours. In order to avoid instabilities due to a negative viscosity, it is required that $\nu_{tot} = \nu + \nu_{sgs} \geq 0$. The tilde $\tilde{\cdot}$, denoting the supra-grid scale quantities, will be omitted henceforth.

2.4.2. The RANS approach

For RANS computations, the standard $k - \epsilon$ model is used in this study for the closure of the Reynolds stress tensor that arises from the time-averaging operation. A good description of the $k - \epsilon$ model can be found in many textbooks on turbulence or computational fluid dynamics. For an extensive overview of the $k - \epsilon$ equations and the associate modelling constants, the reader is therefore referred to, for instance, the book of Pope [24]. The boundary conditions for $k$ and $\epsilon$ are specified at the first grid point at a distance $z_0$ from the wall where the logarithmic law of the wall prevails. With this and the assumption of local equilibrium, there follows $k_w = v_w^2 / \sqrt{\gamma \mu}$ and $\epsilon_w = v_w^3 / \kappa z_0$ (cf. [25]). Only steady RANS calculations are concerned in this paper.

2.5. Boundary conditions

A horizontally stress-free rigid lid is imposed to represent the free surface of the flow. The use of such a rigid lid is justified as long as the gradients of the water surface are small enough. This approach is often used for straight open-channel flows (cf. [26–28] as well as for curved open-channel flows (cf.[9, 10, 25]).

Solid walls are treated with the wall-function approach rather than that the grid is refined to reach the viscous sub-layer near solid walls. For hydraulically smooth solid walls, the standard law of the wall is used with the viscous sub-layer, the buffer layer and the logarithmic layer that incorporated the following:

$$v_n^+ = z_n^+,$$  

if $z_n^+ \leq 5$,

$$v_n^+ = 5.0 \ln z_n^+ - 3.05,$$  

if $5 < z_n^+ < 30$,

$$v_n^+ = 2.5 \ln z_n^+ + 5.5,$$  

if $z_n^+ \geq 30$.  (6)
where \( v_n \) and \( z_n \) represent the wall-normal velocity and coordinate respectively. For hydraulically rough solid walls, a modified log law (cf. [29], p. 100) is used according to

\[
v_n^+ = 2.5 \ln \frac{z_n}{k_s} + 8.5,
\]

where \( k_s \) represents the roughness height. The diameter of the sand particles is related to the roughness height \( k_s \) as three times the sand particle diameter (cf. Van Rijn [16]). Given the diameter of 2 mm of the sand particles, \( k_s \) is chosen to be 6 mm.

In the LES model, the length of the straight inflow and outflow sections are restricted to 3.8 m each in order to save computational costs. A convective boundary condition is used at the outflow. The inflow boundaries are provided by the output of a simulation of the corresponding straight open-channel flow using periodic boundaries in stream-wise direction.

The wall function is applied by calculating the friction velocity from either Equation (6) or (7) and adding the associate shear stress (i.e. the friction velocity squared) to the equations of motion as a surface force. In RANS computations, this shear stress is based on the Reynolds-averaged velocities.

### 2.6. Simulation runs

On the basis of the geometry shown in Figure 1 and the experimental conditions given in Table 1, nine distinct simulations are defined and listed in Table 2. Basically three parameters are varied: the flow depth \( H \), the simulation type (LES or RANS) and the sub-grid model (static with a certain coefficient or dynamic), if applicable.

The labelling of the simulations runs (first column of Table 2) is as follows. The first character denotes the flow depth: S means shallow, M means medium and D means deep. The second character indicates the simulation type: L stands for LES and R stands for RANS. The third character gives information on the used sub-grid model: S stands for a small \( C_s \) value, M for a medium one, L for a large one and D stands for the dynamic Smagorinsky model. As a medium value, the more or less standard value of \( C_s = 0.1 \) is chosen. The other values for \( C_s \) are 0.065 (often used for plane boundary layer flow) and 0.17.

<table>
<thead>
<tr>
<th>Run</th>
<th>( H ) (m)</th>
<th>Simulation type</th>
<th>Sub-grid model</th>
<th>( C_s )</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLM</td>
<td>0.108</td>
<td>LES</td>
<td>static Smagorinsky</td>
<td>0.1</td>
</tr>
<tr>
<td>MLM</td>
<td>0.159</td>
<td>LES</td>
<td>static Smagorinsky</td>
<td>0.1</td>
</tr>
<tr>
<td>DLM</td>
<td>0.206</td>
<td>LES</td>
<td>static Smagorinsky</td>
<td>0.1</td>
</tr>
<tr>
<td>SR</td>
<td>0.108</td>
<td>RANS</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>MR</td>
<td>0.159</td>
<td>RANS</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>DR</td>
<td>0.206</td>
<td>RANS</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>MLS</td>
<td>0.159</td>
<td>LES</td>
<td>static Smagorinsky</td>
<td>0.065</td>
</tr>
<tr>
<td>MLL</td>
<td>0.159</td>
<td>LES</td>
<td>static Smagorinsky</td>
<td>0.17</td>
</tr>
<tr>
<td>MLD</td>
<td>0.159</td>
<td>LES</td>
<td>dynamic Smagorinsky</td>
<td>–</td>
</tr>
</tbody>
</table>
All the simulations are run on a grid containing $1260 \times 192 \times 24$ grid cells in stream-wise, transverse and vertical direction respectively. For convergence analysis, grid independency was tested for run SLM (using another grid containing $1560 \times 200 \times 30$), run MLM (using grids containing $1560 \times 200 \times 30$ cells and $1260 \times 192 \times 36$ cells) and run DLM (using another grid containing $1260 \times 192 \times 48$). Case MR has also been run on a very coarse grid containing $600 \times 96 \times 12 \text{ grid cells}$ showing only small differences in outcome (i.e. velocity differences smaller than 4%). Given these grids, the near wall resolution is found to vary from 20 (finest grid) to 120 wall units (coarsest grid) at the hydraulically rough bottom.

Note that the shallow case ($H = 0.108 \text{ m}$) and the deep case ($H = 0.206 \text{ m}$) slightly differ from their experimental equivalents, since the bulk velocity $V_{av}$ is 0.43 m/s in the simulations, whereas in the experiment the bulk velocity is 0.40 and 0.38 m/s for the respective two cases (see Table 1). However, this does not affect the main parameters $H/R$ and $B/R$ but only the Froude number $Fr$ and Reynolds number $Re$ which do not play a significant role.

3. Main flow characteristics

In this section, an evaluation is made of the performance of both the LES computations and the RANS computations regarding the reproduction of the primary flow characteristics observed in the experiment. Considering the LES computations, the focus is on the case MLM (with $C_s = 0.1$) rather than on the cases with different sub-grid model settings. The sensitivity to changes in the sub-grid model is addressed separately at the end of this section.

3.1. Water levels

Since open-channel flow is driven by gradients in the water surface level, which reflect the energy gradient, the accurate simulation of the water surface topography is a prerequisite for the accurate simulation of the flow field. However, because of the rigid-lid assumption, the pressure is used in the model rather than the water levels. Nonetheless, the time-averaged pressure distribution resulting from the simulations can be used to calculate the equivalent water levels and to evaluate the total loss of energy over the entire flume assuming a hydrostatic pressure distribution in vertical direction. Notwithstanding the non-hydrostatic character of the model and the use of the rigid lid, by this assumption for the free surface, the pressure at the free surface can hence accordingly be translated to an equivalent water level by

$$\frac{\partial h}{\partial r} = \frac{1}{\rho g} \left( \frac{\partial p}{\partial r} \right)_s \quad \text{and} \quad \frac{\partial h}{\partial \theta} = \frac{1}{\rho g} \left( \frac{\partial p}{\partial \theta} \right)_s. \quad (8)$$

In Figure 3, the simulated water surface for run MLM is given. This figure shows the stream-wise decrease of the water surface which is directly related to energy losses due to friction at the walls. In Figure 3, also the transverse super-elevation can be seen which is due to the curvature-induced balance between centripetal forces and pressure forces. At the entry and at the exit of the bend, pronounced gradients in the water surface are found due to the discontinuity in curvature. The highest water level of about 0.168 m can be found at about the $60^\circ$ cross section.
The calculated water levels should be compared with their experimental counterparts to check their validity. In Figure 4, the equivalent water depths are shown for the three cases of Table 1, for the respective 135°, 90° and 75° cross sections. In this figure, it can be seen that the tilting of the water level due to the balance between pressure forces and centripetal forces are reproduced by the models quite accurately. The largest difference is seen at the outer bank of the deep case: the simulated water depth in the LES and RANS is 2.2% higher in this area. The mutual differences between the LES results and the RANS results are rather small.

Moreover, from the pressure distribution we can calculate the total energy losses. Invoking the hydraulic radius \( R_h \),

\[
R_h = \frac{BH}{B + 2H},
\]

(9)

with \( B \) the width and \( H \) the mean water level, we can express the flume-averaged friction factor \( C_f \), defined as

\[
C_f = \frac{\rho \nu^2}{\rho V_{av}^2},
\]

(10)
in terms of the pressure difference $\Delta p$ over the entire flume following:

$$C_f = \frac{\Delta p}{L} \frac{R_h}{\rho \sqrt{\frac{2}{\lambda}}},$$  \hspace{1cm} (11)$$

with $L$ being the length of the flume along the centre line.

The calculated values of $C_f$ from both the LES (with $C_s = 0.1$) and the RANS computations for all three cases (see Table 1) are shown in Table 3 being compared with their experimental equivalents. From this table, it can be deduced that the LES fairly well reproduces the flume-averaged friction losses (differences of about 3%), whereas RANS does not (differences of about 50%), which is a major result with large importance for sound predictions of the conveyance capacity and hence for hazard mapping in case of real-river systems. Taking into account that the LES and RANS computations used the same wall functions, it can be concluded that the turbulence model is the dominant factor in properly reproducing the total friction losses along the flume. For the LES computations, the roughness height $k_s = 6\, \text{mm}$ turns out to be a realistic value to model the hydraulically rough bottom. Moreover, despite the rigid lid assumption, the stream-wise and transverse pressure gradients are well reproduced by the LES computation. Although the RANS computations were able to reproduce a sound water level slope at the locations shown in Figure 4, their overall performance on this issue could be depicted as not adequate.

Table 3. Flume averaged values of $C_f$ from the experiment, the LES (with $C_s = 0.1$) and RANS computations for the three cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>$H$ (m)</th>
<th>$C_f$ from experiment</th>
<th>$C_f$ from LES</th>
<th>$C_f$ from RANS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shallow case</td>
<td>0.108</td>
<td>$5.7 \times 10^{-3}$</td>
<td>$5.9 \times 10^{-3}$</td>
<td>$8.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Medium case</td>
<td>0.159</td>
<td>$5.8 \times 10^{-3}$</td>
<td>$6.0 \times 10^{-3}$</td>
<td>$8.9 \times 10^{-3}$</td>
</tr>
<tr>
<td>Deep case</td>
<td>0.206</td>
<td>$5.7 \times 10^{-3}$</td>
<td>$5.8 \times 10^{-3}$</td>
<td>$7.6 \times 10^{-3}$</td>
</tr>
</tbody>
</table>
Figure 5. Depth-averaged stream-wise velocities, scaled with the bulk velocity $V_{av}$, for the experiment (a), the LES computation (b) and for the RANS computations (c). The medium case is shown, i.e. the water depth $H = 0.159$ m. The flow enters the picture at the lower right corner.

3.2. Velocity distribution

In Figure 5, the time-averaged stream-wise velocities averaged over the flow depth are shown for the experiment, the LES computation and the RANS computation of the medium case ($H = 0.159$ m). The picture of the experimental result is obtained by spreading the data of a certain cross section over its close vicinity. In this figure, it is visualised how turbulent channel flow enters the bend and how this flow is subjected to centripetal forces in the curved part of the geometry. A clear outward shift of momentum is identified. In the curved part of the flume, the centripetal forces give rise to secondary flow cells. At the entry of the bend, the highest velocities are found near the inner bank of the flume. This is explained by the suddenly strong favourable pressure gradient at the inner bank.

The most pronounced difference between the three pictures in Figure 5 is seen in the behaviour of the flow near the convex inner bank of the flume. It is observed for the experiment as well as for the LES computation that the main flow detaches from the inner bank whereas the flow in the RANS computation remains attached. Since for both the LES computation and the RANS computation the same wall functions are used, this dissimilar behaviour can be attributed to the turbulence model and the unsteadiness of the LES computation. Since the pictures of the experimental result and the LES result do not exhibit return flow, the detachment of the main flow from the convex inner bank will henceforth be referred to as an internal shear layer (cf. Simpson [4]).
For a further and more quantitative investigation of the flow field, we highlight the profiles of the stream-wise and transverse velocities at some cross sections in the flow. For the shallow case, the 135° cross section and for the deep case the 75° cross section is considered, as for these two cases only one cross section has been measured. For the medium case, we consider the 90° cross section. In addition, the 180° cross section is highlighted in order to provide a look further at the downstream development. The transverse locations of the verticals are 0.10B, 0.17B, 0.27B, 0.46B, 0.65B, 0.79B and 0.88B. The results for the time-averaged stream-wise velocities $V$ and the time-averaged transverse velocities $U$, both non-dimensionalised using the bulk velocity $V_{av}$, are shown in Figures 6 and 7 respectively.

The velocity profiles from the experiment and the LES quite reasonably coincide for the shallow case, as can be seen from Figures 6 and 7. The transverse velocities seem to be underestimated by the RANS computation in the core of the cross section. The agreement of the velocity profiles for the medium case is also quite well, especially for the 180° cross section. At some particular profiles, discrepancies are visible. Moreover, compared with the
shallow case, an underestimation of the transverse velocities is seen for the results of the RANS computation. This underestimation has also been found by Zeng et al. in the results of their RANS simulation (with a one-equation closure model) of the same experiment. Considering that the transverse velocities are properly reproduced by the present LES, it can be concluded that the choice of the turbulence modelling approach considerably influences the strength of the secondary circulation.

For the deep case, the agreement between the experimental and numerical results are of lower quality compared with the shallow and medium case. The transverse velocities seem to be overestimated by the LES in the middle area of the cross section. The disagreement might be due to the difference in the bulk velocities of the flow in the experiment (0.38 m/s) and the LES (0.43 m/s), but could also be due to the rigid-lid assumption for the free surface as it can be seen in Figure 3 (although for the medium case) that the transverse gradients are the largest in the upstream part of the bend where the flow is strongly spatially developing. An alternative explanation is the reduced accuracy of the measurement device near the free surface.
Figure 8. Vertical velocities at the 75° cross section of the deep case. (a) Experimental result; (b) LES results. The values are made non-dimensional using the bulk velocity $V_{av} = 0.43 \text{ m/s}$. From left to right denotes from inner bank to outer bank.

Because of this weaker agreement of the stream-wise and transverse velocities for the deep case, the structure of the vertical velocities is shown in Figure 8 in addition. This figure shows the experimental pattern in the Figure 8(a) and the pattern obtained by an LES computation on a grid with 24 cells in the vertical direction [Figure 8(b)] and an LES computation on a grid with 48 cells in the vertical direction. In this figure it can be seen that the structure of the vertical velocities is reproduced quite well by the LES, fairly independently of the chosen mesh. An interesting aspect of Figure 8 is that it nicely reveals the complex structure of the secondary flow. In the left part of the cross section (i.e. near the inner bank) an alternating pattern of upwelling and downwelling is observed. This pattern is associated with the internal shear layer separating from the inner bank.

The flow in the right part of the cross section (i.e. near the outer bank) is obviously dominated by downwelling associated with the main secondary flow motion due to the presence of the centripetal forces. However, a small region of upwelling flow is identified at the outer bank, which is associated with a secondary flow cell with an opposite rotational sense. This counter-rotating secondary flow cell will henceforth be referred to as the outer-bank cell and will be discussed in more detail in the following of this paper.

A better view on the structure of the cross section is offered by the stream-wise vorticity $\omega_\theta$, defined as

$$ \omega_\theta = \frac{\partial W}{\partial r} - \frac{\partial U}{\partial z} $$

(12)

which renders the rotational strength of the secondary flow. In addition to the vertical velocities for the experiment and the two LES computations of the deep case using 24 and 48 cells in the vertical direction, respectively (shown in Figure 8), the stream-wise vorticity is shown for the deep case in Figure 9 to investigate the similarities between the
Figure 9. Streamwise vorticity at the 75° cross section of the deep case. (a) Experimental result; (b) LES results. The values are made non-dimensional using the mean water depth $H$ and the bulk velocity $V_{av} = 0.43$ m/s. From left to right denotes from inner bank to outer bank.

Experimental approach and LES approach. Figure 9 shows that the pattern is quite similar. However, the values for the vorticity are somewhat higher in the outer-bank region and somewhat lower in the core region of the cross section in the experimental results compared with the LES results.

In the inner half of the cross section, the internal shear layer is identified by the existence of a belt of positive vorticity values at the free surface. This belt is more pronounced in the LES result compared with the experimental result. It should however be kept in mind that in the upper 10% of the water depth the experimental results are not reliable. The RANS are kept out of consideration completely in Figure 9 since it completely lacks the internal shear layer as well as the outer-bank cell. Therefore, a comparison of the vorticity is not very useful.

3.4. Velocity profiles along the centre line

In Figure 10, the vertical profiles of the stream-wise velocities at the centre of the cross sections at 15°, 30°, 60°, 90°, 120°, 150° and 180° in the bend are shown for the shallow case, the medium case and the deep case. In this figure, the solid lines represent LES data obtained with a grid containing $1260 \times 192 \times 24$ cells. The dashed lines represent LES data obtained with a grid containing $1580 \times 200 \times 30$ for the shallow case, $1260 \times 192 \times 36$ for the medium case and $1260 \times 192 \times 48$ for the deep case. Grid changes for the medium case and the deep case are merely focussed on refinement in the important vertical direction.

Figure 10 shows that the agreement of the numerical data with the experimental data is generally fairly good. Moreover, it shows that the numerical data are found to be independent
of the chosen mesh. Figure 10 furthermore shows that the vertical location of the maximum velocity moves towards the bottom with increasing water depth, which is typical for strongly curved flows. This tendency is found to be somewhat stronger in the LES data compared with the experimental data.

3.5. Sensitivity to changes in the sub-grid model

In order to investigate the sensitivity of the results to changes in the sub-grid model, the vertical vorticity is highlighted. The vertical vorticity $\omega_z$, defined as

$$\omega_z = \frac{1}{r} \frac{\partial u}{\partial \theta} - \frac{1}{r} \frac{\partial vr}{\partial r},$$

provides a sound insight in the physics of the flow along the bend. It is for this reason that the vertical vorticity is considered for the four cases with different sub-grid model settings in Figure 11. Visualised at the free surface, the vertical vorticity (shown for the cases MLS, MLM, MLL and MLD, i.e. for the cases with medium depth) exhibits the flow structure over the key part of the flume. For the first three cases, the static Smagorinsky model is used with different values ($C_s = 0.065, 0.1$ and $0.17$), whereas for the latter case, the dynamic Smagorinsky model is used. This figure nicely reveals the rotational structure at the free surface and clearly shows the detachment of the boundary shear layer at the inner bank.

The figure shows that the cases with $C_s = 0.065$ and $C_s = 0.1$ show very similar results. Differences between these two cases on the one hand and the case with the dynamic
Figure 11. Vertical vorticity at the free surface for the medium case obtained with different sub-grid models. In lexicographic ordering: static Smagorinsky with $C_s = 0.065$, static Smagorinsky with $C_s = 0.1$, static Smagorinsky with $C_s = 0.17$ and dynamic Smagorinsky. The flow enters the picture at the lower right corner.

Smagorinsky model on the other hand are also of minor importance, apart from the observation that the peak values of the vorticity in the shear layer that detaches from the convex inner bank are somewhat lower if the dynamic Smagorinsky model is used. Moreover, the width of the shear layer is somewhat wider for the case with the dynamic Smagorinsky model. The region near the concave outer bank is very similar for the case with the dynamic Smagorinsky model compared with the case with $C_s = 0.1$. The case with $C_s = 0.17$ shows some dissimilarities compared with the three other cases: the point of detachment of the internal shear layer near the inner bank is located more upstream compared with the three other cases. Moreover, in the outer bank region upstream in the bend, the vorticity is lower compared with the values in the other three cases and even changes sign.

After all, it can be concluded that the changes in the sub-grid model do not effectuate major changes in the flow field. This conclusion subsequently raises the question why this is the case. In order to answer this question, the balance equation for stream-wise momentum is considered term-by-term in the core region of the $90^\circ$ cross section. This equation is given as

$$ -\frac{\partial V}{\partial t} = U \frac{\partial V}{\partial r} + V \frac{\partial V}{\partial \theta} + W \frac{\partial V}{\partial z} + \frac{UV}{r} + \frac{1}{r} \frac{\partial P}{\partial \theta} $$

$$ + \frac{\partial u'v'}{\partial r} + \frac{1}{r} \frac{\partial v'v'}{\partial \theta} + \frac{\partial v'w'}{\partial z} + \frac{2}{r} u'v' + \text{DIFF}, $$

(15)
in which the diffusion term \( \text{DIFF} \) reads as

\[
\text{DIFF} = \nu_{\text{tot}} \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2} \right) \left( \frac{V}{r^2} + \frac{2}{r} \frac{\partial U}{\partial \theta} \right). \tag{16}
\]

Capitals denote time-averaged velocities. From the analysis of the stream-wise momentum balance equation, it was found that some terms are of negligible importance in the core region of the flow, namely the terms:

\[
W \frac{\partial V}{\partial z}, \quad \frac{\partial \overline{u'v'}}{\partial r}, \quad \frac{1}{r} \frac{\partial \overline{v'v'}}{\partial \theta} \quad \text{and} \quad 2 \frac{\overline{u'v'}}{r}. \tag{17}
\]

The remaining six terms are shown for the core region of the 90° cross section (in the middle of the bend) of the medium case (run MLM) in Figure 12. The core region of the 90° cross section was found to be very similar to the core region in other parts of the bend as long as the stream-wise momentum equation is concerned. In Figure 12, it can be seen that the turbulence term and the \( \text{DIFF} \) term only play a role near the bottom, whereas in the major part of the cross section, the flow is primarily dominated by the centripetal term \( U/V/r \) and secondarily by advection and stream-wise pressure gradients. This, thereby, indicates that the closure of the Reynolds stress tensor is only of minor importance in this area.

If turbulent momentum transport is indeed of minor importance with respect to the distribution of the stream-wise velocities, and if, moreover, the sub-grid modelling of turbulence is of minor importance in LES computations, one might question why the RANS results are so poor. The difference between RANS at one hand and LES at the other hand and the importance of the turbulence can be explained as follows. Adective momentum transport by the secondary flow cell in the centre of the cross section and the internal shear layer near the inner-bank and the outer-bank cell was found to be a dominant mechanism with respect to the velocity distribution. These secondary flow cells are largely induced by turbulence, as indicated by the fact that they can be resolved by LES models but not by RANS models (with an isotropic turbulence model). The fact that the LES models can adequately resolve these flow features, more or less irrespective of the sub-grid modelling, indicates that the essential role is played by the large scales of turbulence, which interact with the mean secondary flow structures, whereas the small-scale turbulence is merely dissipative and does not play a dynamic role with respect to the time-averaged flow pattern (cf. [30]).

4. Bed shear stresses

A key element in the study of natural river flows is the distribution of the bed shear stresses. The bed shear stresses are a measure of the forces that are exerted on the bed and is therefore important for the prediction of erosion and sedimentation processes. If the bed shear stress exceeds the critical shear stress, then motion of the grains situated at the bed is initiated. Note that most sediment transport formulae describe bed load sediment transport to be roughly proportional to the bed shear stress to the power 1.5, implying that inaccurate predictions in the bed shear stress will be amplified in inaccurate predictions of sediment transport and bathymetry.
The bed shear stresses can simply be extracted from the simulations. Given the hydraulically roughness of the bed, Equation (7) can directly be applied to obtain the friction velocity. The bed shear stress is just the squared friction velocity. We represent the thus calculated bed shear stresses by the friction factor $c_f$, which is defined as

$$c_f = \frac{\rho v^*_v^2}{\rho V_{av}^2},$$

in which $v^*_v$ is the friction velocity. The wall function is applied on the total velocity vector, based on the two wall-parallel velocity components. Note that the spatially averaged value of $c_f$ equals the overall friction factor $C_f$, defined in Equation (11).

The spatial distributions of the friction factor $c_f$ in the bend, as these result from the LES computation and the RANS computation, are shown in Figure 13 for the medium case. In this figure, it can be seen that the computed bed shear stresses from the RANS computation are significantly higher than from the LES computation. In the LES case, the bed shear stresses near the inner bank vary from about $8 \times 10^{-3}$ to $10 \times 10^{-3}$, whereas in the RANS case, the bed shear stresses in this area even exceed $12 \times 10^{-3}$. 

Figure 12. Several budgets of the stream-wise momentum balance equation in the core region of the 90° cross section of the medium case (run MLM). The values are made non-dimensional by the water depth $H$ and the bulk velocity squared $V_{av}^2$ and multiplied by $10^3$. 
Figure 13. Friction factor $c_f$ (multiplied by $10^3$) for the medium case. (a) LES results (case MLM); (b) RANS results (case MR). The flow enters each picture at the southeast corner of the picture.

In Section 3.1, we have already seen that the RANS computation does not reproduce the overall friction losses properly, whereas LES does (see Table 3). Figure 13 shows that the poor reproduction of the friction losses can be explained by the overestimation of the bed shear stresses. If Figure 13 is connected with Figure 5, it can be understood that the overestimation of the bed shear stresses is due to the incorrect prediction of the main flow field. The relation between the main flow field and the bed shear stresses hence emphasises the eminent importance of the proper reproduction of the flow field and thereby the importance of the choice of the turbulence model.

5. Secondary flow pattern

Since secondary flow is the key element of a curved open-channel flow, a detailed analysis of the secondary flow structure is of eminent importance. In this section, we investigate the secondary flow features with the focus on its development along the bend, the influence of the turbulence model (LES versus RANS), the interpretation of the outer bank cell and the stream-wise development of the internal shear layer near the inner bank. These processes play an important role in natural open-channel bends, for example with respect to protection of the outer bank against erosion, erosion of the outer bank, accretion at the inner bank and the river planform evolution.

5.1. Structure of the cross section

In Figure 14, the distribution of the stream-wise vorticity is shown for the $90^\circ$ cross section of the medium case. From top to bottom are shown: the results from the experiment, the LES computation on the standard grid, the LES computation on a grid refined in the vertical direction and the RANS computation. The experiment and the LES results show a centre region cell (the major area of negative vorticity), an outer-bank cell (an area of positive vorticity near the outer bank) and a internal shear layer near the inner bank (an area of positive vorticity near the inner bank). Qualitatively, the LES results agree well with the experimental data. The RANS computation does, however, not reproduce the outer-bank cell and the internal shear layer at all. We will discuss these three areas of the cross section in detail in the following section.
Figure 14. Non-dimensional streamwise vorticity distribution at the 90° cross section of the medium case. From top to bottom: the results from the experiment, the LES computation (MLM) on the standard grid, the LES computation on a grid refined in the vertical direction and the RANS computation. From left to right denotes from inner bank to outer bank.

5.2. The centre region

In order to more quantitatively analyse the obtained results, we call upon the circulation $\Gamma$, defined as:

$$\Gamma = \iiint_{A_c} \omega_{\theta} \, dA_c,$$

which is a measure of the strength of a certain vortex tube. To enable proper comparison of the strength of the several vortex tubes, which the centre region cell and the outer-bank cell basically are, the circulation $\Gamma$ is scaled as follows:

$$\hat{\Gamma} = \frac{1}{A_c} \frac{H}{V_{av}} \Gamma,$$

in which $A_c$ represents the area of the vortex tube at a certain cross section. Given the definition of the vorticity $\omega_{\theta}$ given in Equation (12), the centre region cell has a negative circulation and the outer-bank cell a positive circulation.

The circulation $\hat{\Gamma}$ of the centre region cell is given for the three cases (experiment and LES and RANS computations) in Figure 15 by the black lines. As regards the LES, only the cases with $C_s = 0.1$ are shown; it was found for the medium case that the changes in the sub-grid model settings only marginally affect the results. In Figure 15, it can be seen
that the experimental and LES values of the circulation fairly well agree, but also that in the region of strong spatial development of the flow (say, from $30^\circ$ to $80^\circ$) the LES data overestimate the measured circulation, which has readily been discussed previously. The RANS computation, on the contrary, underestimates the measured data along the entire curved reach of the flume. This underestimation can be explained by the absence of the internal shear layer near the inner-bank and the outer-bank cell, which enables the centre region cell to cover the entire cross section. Figure 15 shows that the strength of the centre region cell increases with increasing water depth in the upstream part of the bend.

For all the three cases, the circulation grows considerably upon entering the bend, reaches a maximum value between $60^\circ$ and $90^\circ$, and subsequently decays towards a similar value for all the three cases. This behaviour is not in agreement with common parameterisation that predict the secondary flow and the corresponding circulation to increase proportionally with the flow depth for a given radius of curvature (e.g. [31–33]). However, Blanckaert
& De Vriend [34] have successfully resolved these tendencies by means of a 1-D model with simple turbulence closure that takes into account the nonlinear feedback between the stream-wise flow and the circulation. Although the qualitative pattern of the secondary flow and the circulation can be reproduced by means of a simple turbulence model, the differences between the LES and RANS computations indicate that turbulence plays an important role with respect to the accurate quantitative reproduction.

5.3. The outer-bank region

Figure 15 also shows that the dependency of the outer bank cell strength on the water depth is similar to that dependency for the centre region cell: it increases with increasing water depth. The results for the medium case show though that the outer bank cell is the strongest in the upstream part of the bend, both in the LES results and in the experimental results, whereas it loses strength in the downstream part of the bend. Considering its sensitivity to scatter of the data, the agreement of the LES data with the experimental data in the upstream part of the bend can be depicted as fair. However, the outer bank cell vanishes at 70° in the bend for the shallow case, at 110° for the medium case and at 160° for the deep case, whereas the outer bank cell is present in the experimental results for all the three cases. Why this cell disappears at some distance in the LES is not really clear. But, the outer bank cell reappears if a very long bend is simulated using periodic boundaries in stream-wise direction (not shown here). In the RANS results, the outer bank cell is no where seen and therefore not plotted in Figure 15.

If we define $L_{obc} = \sqrt{H_{obc} \times B_{obc}}$ as a characteristic length scale of the outer bank cell (with $H_{obc}$ and $B_{obc}$ being the typical height and width of the cell), then we find in both the experimental results and the LES results that $L_{obc} \approx 0.8H$ in the upstream part of the bend, and for the experimental results solely $L_{obc} \approx 0.5H$ in the downstream part of the bend. Apparently, the decrease of both the strength and the size of the outer bank cell is overestimated by the LES. This observations was also found to hold for the cases with different sub-grid model settings; changes in the sub-grid model did not yield considerable quantitative differences.

The outer bank cell has received quite some attention recently. It has been observed in the laboratory experiments of, for instance, Booij [9] and Blanckaert and De Vriend [30] and in a real natural river system by Bathurst et al. [35]. Similar to Blanckaert and De Vriend [30] and Van Balen et al. [11], the mechanisms underlying the outer-bank cell will be investigated by means of an analysis of the sources and sinks of the balance equation for stream-wise vorticity. Whereas Van Balen et al. [11] could neglect gradients in stream-wise direction thanks to the mild curvature of the flow, these terms have to be accounted for in the present sharp bend.

In order to better understand the physics behind the outer bank cell, the balance equation for the stream-wise vorticity is subjected to a term-by-term analysis. For this purpose, the cross section at 40° of the medium case is chosen as at this location the circulation of the outer bank cell peaks. This balance equation for the stream-wise vorticity $\omega_\theta$ [defined by Equation (12)] reads

$$\frac{\partial \omega_\theta}{\partial t} = ADV + CFG + ISO + HOM + SKW + DNU + DIFF,$$

(21)
with

\[ ADV = - \left( U \frac{\partial \omega}{\partial r} + \frac{V}{r} \frac{\partial \omega}{\partial \theta} + W \frac{\partial \omega}{\partial z} \right), \]

\[ CFG = - \frac{1}{r} \frac{\partial}{\partial z} \left( V^2 + \frac{u'^2}{\omega^2} \right), \]

\[ ISO = \frac{\partial^2}{\partial r \partial z} \left( U^2 - W^2 \right) + \frac{1}{r} \frac{\partial u'^2}{\partial z}, \]

\[ HOM = \left( \frac{\partial^2}{\partial z^2} - \frac{1}{r} \frac{\partial}{\partial r} \frac{1}{r} \frac{\partial}{\partial r} + \frac{1}{r^2} \right) u'w', \]

\[ SKW = \frac{U}{r} \frac{\partial W}{\partial r} + \frac{U}{r} \frac{\partial U}{\partial z} + \frac{U}{r} \frac{\partial V}{\partial z} + \frac{V}{r^2} \frac{\partial W}{\partial \theta} + \frac{V}{r} \frac{\partial U}{\partial \theta}, \]

\[ DNU = \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial u'^2}{\partial \theta} \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left( \frac{1}{r} \frac{\partial v'^2}{\partial \theta} \right) + \frac{\omega}{r} \frac{\partial V}{\partial \theta}, \]

\[ DIFF = \nu_{tot} \left( \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2} - \frac{1}{r^2} \right) \omega \]

\[ + \nu_{tot} \frac{2}{r^2} \frac{\partial}{\partial \theta} \left( \frac{\partial V}{\partial z} - \frac{1}{r} \frac{\partial W}{\partial \theta} \right), \]

in which \( ADV \) represents advection of \( \omega \), \( CFG \) the centrifugal effects, \( ISO \) the anisotropy of the turbulence stresses, \( HOM \) the inhomogeneity of the turbulence stresses, \( SKW \) the redistribution of \( \omega \) by skewness of the velocity field excluding terms that can be associated with centrifugal effects, \( DNU \) the downstream non-uniformity of the turbulence stresses including vortex stretching (last term of \( DNU \)) and \( DIFF \) the generation of vorticity by viscous effects. The combined effects of the molecular viscosity \( \nu_{mol} \) and the sub-grid viscosity \( \nu_{sgs} \) are captured by \( \nu_{tot} \).

The separate budgets of Equation (21) are shown in Figure 16 for the 40° cross section of the medium case. In this figure, it can be seen that the terms \( ADV \), \( CFG \) and \( ISO \) are clearly the most relevant budget terms of the vorticity equation. The pictures of the terms \( HOM \), \( SKW \) and \( DNU \) show generally much lower values and do not show a pattern that could be related to the rotational structure in a meaningful way. Notice that the advection term \( ADV \) contains a component in stream-wise direction, representing inertial adaptation of the secondary flow in stream-wise direction.

It can be seen in Figure 16 that a clear relation is present between the vorticity and the \( CFG \) term. Since these terms have the same sign in the outer bank area, it can be stated that centrifugal effects favour the rotational sense of the flow. It was found that the turbulence contribution to the \( CFG \) term is much smaller than the mean flow contribution, i.e.

\[ CFG \approx - \frac{1}{r} \frac{\partial V^2}{\partial z}. \]  

The \( ADV \) term and the \( ISO \) term seem to be opposite to each other and both terms change sign at the location at the outer bank where \( w \) changes sign and at the location at the free surface where \( u \) changes sign.
Based on Figures 16, the following interpretation of the outer bank cell can be formulated. It is known that in corners of the flow geometry, turbulence anisotropy gives rise to distortion of the profile of the stream-wise velocities (cf. [28, 36, 37]). Due to this distortion, the CFG term comes to existence via Equation (22), generating vorticity that is mostly counteracting to the ADV term, shows that turbulence anisotropy still plays a major role once the outer bank cell has come into existence (cf. [11, 30]).

The existence of coherent structures along concave walls, so-called Taylor–Görtler vortices, have amply been studied for closed curved ducts in the past (see [2, 3, 7]). Basically, the outer bank cell could be depicted as a Taylor–Görtler vortex along the concave outer...
bank. In fact, the present study illustrates how such a pronounced Taylor–Görtler vortex exists under the presence of a free surface and how it can be explained from a term-by-term analysis of the balance equation for stream-wise vorticity. The present findings are in line with the experimental and computational results found in literature (cf. [9, 11, 30]). The validity of the present findings are not particularly dependent on the corner eddies as found in the straight upstream reach of the flow geometry as the results of an analysis of the outer bank cell in a very long bend with periodic boundary conditions (without straight inflow reach) is found to be similar to the results of the present analysis (cf. [11]).

5.4. The inner bank region

As shown in several figures above (e.g. Figures 5, 9, 11), an internal shear layer comes into existence at the convex inner bank of the bend due to the strong curvature of the flow and the associated upwelling of low-momentum fluid. Note that the flow is not recirculating as stream-wise velocities remain positive. The influence of this internal shear layer increases in strength with increasing water depth. The location of detachment of this shear layer from the inner bank is at about 30°, 40° and 45° in the bend for the shallow, medium and deep case respectively.

In Figure 17, some cross-sectional distributions of the stream-wise velocities are shown along the curved part of the deep case in order to illustrate how the resulting dip in the stream-wise velocities evolves throughout the flow. The cross sections are located from the 25° cross section to the 178° cross section with intervals of 17° in the curved part of the flume.

Figure 17. Pattern of the time-averaged stream-wise velocities at several cross sections of the deep case. The arrow denotes the flow direction. Distances are in metres, velocities are in metres per second. The cross sections are located at 25° : 17° : 178° in the curved part of the flume.
Figure 18. Pattern of the stream-wise gradients of the time-averaged pressure at several cross sections of the deep case. The arrow denotes the flow direction. Distances are in metres, pressure gradients are in metres per second square. The cross sections are located at $25^\circ : 17^\circ : 178^\circ$ in the curved part of the flume.

In addition, patterns of the stream-wise pressure gradient, $\partial p/\partial \theta$, are shown in Figure 18. An adverse pressure gradient comes into existence along the inner bank of the flume (second and third upstream cross sections). An associated local minimum in the stream-wise velocity profile seems to be advected by the helical motion and, moreover, is amplified by an adverse pressure gradient.

The adverse stream-wise pressure gradient gains influence near the inner bank about halfway in the bend, where the local minimum in the stream-wise velocities is most pronouncedly seen. In the downstream part of the bend, the adverse pressure gradient is intensified by the sudden transition from curved flow to straight flow at the exit of the bend. The shear layer is accompanied by downwelling motion somewhat away from the inner bank (see Figure 8 at $x/H \approx 1$).

6. Turbulence structure

In the previous section, we have seen that the choice for either the direct approach (LES) or the statistical approach (RANS) has eminent consequences as regards the sound reproduction of the internal shear layer near the inner bank and the counter-rotating cell near the outer bank. Therefore, an investigation of the turbulence stresses near the corners of the flow is worthwhile. Moreover, the structure of the Reynolds stress tensor components can be put in perspective by comparison with straight open-channel flow.
Figure 19. Streamwise-vertical Reynolds stresses $\bar{v}'\bar{w}'$ at the 60$^\circ$ cross section of the medium case. (a) Experimental result; (b) LES result; (c) RANS result. The values are made non-dimensional using the bulk velocity squared $V_{av}^2$. From left to right denotes from inner bank to outer bank.

6.1. Turbulence stresses in the centre region

Results for the streamwise-vertical $v'w'$ stresses over an entire cross section of the medium case are shown in Figure 19 and results for the stream-wise-transverse $u'v'$ stresses are shown in Figure 20. The comparison is restricted to the 60$^\circ$ cross section, since this is the cross section closest to the location where the centre region cell strength peaks in the LES results and where experimental data are available. Notice that the strength of the centre region cell does not differ so much at 60$^\circ$ compared with at 90$^\circ$ in the experiment.

In Figure 19, it can be seen that the $v'w'$ stresses from the LES are slightly higher (in the absolute sense) than those from the experiment, which might be explained by the estimated uncertainty of 15% in the experiment and by the fact that the vertical velocity fluctuations are systematically underestimated by the ADVP measurement device. Nonetheless, the agreement of the structure of these stresses is quite good as far as the experiment and the LES computation are concerned. The $v'w'$ stresses from the RANS computation are lower compared with the results from the LES computation and do not well reproduce the correct pattern near the sidewalls, which can be understood from the results of the entire flow field.

The results for the $u'v'$ stresses, shown in Figure 20, are also of satisfactory quality: the patterns from the experiment are quite well reproduced by the LES, with values of similar magnitude. A clear difference is, however, seen for the $u'v'$ stresses at the 60$^\circ$ cross section: at $x/H \approx 7$ an area of high values is observed near the free surface which is not present in the experimental results. Such an area of high $u'v'$ stress values was also observed in the measurements of Booij [9] and the computations of Van Balen et al. [11] and is associated with the touching between the centre region cell and the outer bank cell. The reason why
Figure 20. Streamwise-transverse Reynolds stresses \( \overline{u'v'} \) at the 60° cross section of the medium case. (a) Experimental result; (b) LES result; (c) RANS result. The values are made non-dimensional using the bulk velocity squared \( V_{av}^2 \). From left to right denotes from inner bank to outer bank.

the experimental result do not show this area might be explained by the fact that about in the upper 15% of the water column the flow is perturbed by the measuring device.

The RANS computation seriously fails in reproducing the correct \( \overline{u'v'} \) stresses. This shortcoming of RANS computations that make use of an isotropic eddy viscosity model to correctly compute the \( \overline{u'v'} \) stresses has already been addressed by Booij [9]. Nonetheless, Figure 12 has shown for the core region of the bend that these stresses only play a negligible role in the transfer of stream-wise momentum. Only the \( \overline{v'w'} \) stresses are of some importance in this respect. Therefore, it could be doubted whether the incorrect closure of the \( \overline{u'v'} \) stresses with an isotropic eddy viscosity model is a serious problem in the core region of the bend. However, near the inner bank, the underestimated values of the turbulence stresses and the thereby underestimated role of turbulence does have serious consequences as readily highlighted previously by the misrepresented detachment behaviour of the boundary shear layer (see Figure 5).

It was found for all the six Reynolds stress components that the RANS values are generally lower than the experimental and LES values. This underestimation of the Reynolds stresses by RANS might explain the underestimation of the circulation (shown in Figure 15). This figure has shown that the circulation as predicted by RANS is about 20%–50% lower compared with LES. Since it was shown by Equation (21) and Figure 16 that turbulence stresses play a pronounced role as a source/sink of stream-wise vorticity, the underestimation of the turbulence stresses does also have consequences for the pattern and magnitude of the stream-wise vorticity. Thus, this underlines, once again, the importance of the choice for an LES approach or a RANS approach.
6.2. Turbulence stresses in the outer bank region

The structure of the Reynolds stress tensor components is shown in Figure 21 for the 40° cross section of the medium case. This 40° cross section is shown to be the cross section where the strength of the outer bank cell has a maximum (see Figure 15).

A similar investigation of the Reynolds stress tensor of the open-channel flow near a corner of the geometry has been carried out by Broglia et al. [28] in case of straight flow. The basic difference is that the current outer bank cell is much stronger than the corner eddy occurring in straight open-channel flow. Figure 21 clearly reveals how the outer bank cell leaves a pronounced footprint in the pattern of the Reynolds stress components.

The stresses $u'u'$, $v'v'$ and $w'w'$ show a strong local increase at the location at the free surface where the outer bank cell and the centre region cell touch. The $w'w'$ stresses do not show this behaviour, since at the free surface $w = 0$ and thereby $u'u' = v'v' = w'w' = 0$. The transverse-vertical $u'v'$ stresses reveal a strong antisymmetry along the bisector of the corner formed by the water surface and the outer bank. Similar patterns were observed in curved open-channel flow over a mobile bed (cf. [38]) as well as in straight open-channel flow over flat bed (cf. [28]). The $v'w'$ stresses show a clear correlation with the rotational sense of the cross-sectional flow, a correlation that is also visible in Figure 19. This correlation is also found for straight open-channel flows (cf. [28]).
In order to render the turbulence anisotropy and the efficiency in turbulence shear stress production, the principal stresses and the so-called structure parameter \( a_1 \) (cf. [39]) are addressed now. The principal stresses are denoted as

\[
\sigma_{1,2} = \frac{\overline{u'^2} + \overline{w'^2}}{2} \pm \sqrt{\left(\frac{\overline{u'^2} - \overline{w'^2}}{2}\right)^2 + \overline{u'w'^2}}.
\]

(23)

Basically, \( \sigma_1 - \sigma_2 \) is the indicator of turbulence anisotropy in the cross-sectional plane, in view of Mohr's circle. Furthermore, the structure parameter \( a_1 \) is defined as

\[
a_1 = \frac{\sqrt{\overline{u'w'^2} + \overline{v'w'^2}}}{2k},
\]

(24)

with \( k = \frac{1}{2} \overline{u'v'} + \frac{1}{2} \overline{v'^2} + \frac{1}{2} \overline{w'^2} \). It can be shown for the case of straight open-channel flow that this non-dimensional parameter \( a_1 \) is zero at the free-surface, has a maximum of 0.14 almost halfway the water depth and a value of 0.10 at the bottom. According to Schwarz & Bradshaw [39], this structure parameter \( a_1 \) roughly indicates the efficiency of turbulent eddies in producing turbulence shear stresses. Basically, the structure parameter is the norm of two elements of the normalised anisotropy tensor (cf. [11, 24]).

Figure 22 clearly shows that the transverse and vertical velocity fluctuations are constrained by the geometry. At the free surface, vertical velocity fluctuations tend to zero, which causes the transverse velocity fluctuations to be dominant. At the outer bank, it is the other way around. As a result, the anisotropy is the most pronounced at the boundaries. The pattern of the principal stress difference \( \sigma_1 - \sigma_2 \) shows, once again, the strong increase of turbulence activity at the touching of the centre region cell and the outer bank cell at the free surface. At this location, the turbulence anisotropy is very high.

For the structure parameter \( a_1 \), it is observed that along the interface of the two mutually counter-rotating cells the values of \( a_1 \) are relatively low. Moreover, it is seen that the decrease of \( a_1 \) at the touching of the two cells at the free surface is accompanied by a strong increase of the \( \overline{u'v'} \) stresses, whereas the \( \overline{u'v'} \) and \( \overline{v'w'} \) stresses are relatively low in this area. In the core of the outer bank cell, however, this dependency is the opposite. It is thereby clear that the outer bank cell itself leads to a locally enhanced production of turbulent shear stress in the cross-sectional plane.

Figure 22. Principal stress difference (a) and structure parameter (b) in the area near the outer bank of the 40° cross section. The data are scaled with the bulk velocity squared \( V_{av}^2 \) and multiplied by 10³.
7. **Summary and conclusions**

In the present paper, results are shown of experiments, LES computations and RANS computations of turbulent flows through a strongly curved single-bend open channel. The simulations were based on the experimental setup of Blanckaert [15] and provide a complementary data set in order to acquire a complete picture of this type of flow. The flow through the experimental facility was studied for three different water depths while keeping the bulk velocity the same for each case. The paper focuses on the main flow, the secondary flow and the role of turbulence.

In order to summarise the findings acquired in the present study, the five research questions postulated in the introduction are recapitulated one by one. Firstly, what is the role of turbulence regarding flow processes in curved open-channel flow? Regarding the transport of stream-wise momentum in the core region of the flow, it was observed that only the stream-wise-vertical Reynolds stress component plays a role, whereas the contributions of the other components are negligible. As regards the outer bank cell, the proper reproduction of the proper turbulence structure has shown to be of eminent importance for the reproduction of the outer bank cell. After a detailed term-by-term analysis of the balance equation for stream-wise vorticity, it turned out that the outer bank cell, basically a Taylor–Görtler vortex, is the net result of the non-linear interplay of turbulence anisotropy and centripetal effects acting on the thus deformed pattern of the stream-wise velocities.

Secondly, what is the performance of LES- and RANS-type computations regarding the simulation of curved open-channel flows? It was found that, generally, LES yields much better results than RANS. Five aspects are important in this respect:

1. LES properly reproduces the experimentally measured overall friction losses over the bend, whereas RANS overestimates these losses. For real rivers, this has considerable consequences as regards the prediction of the conveyance capacity and hazard mapping.
2. The centre region cell strength is underestimated by the RANS model and fairly well estimated by the LES model. This has important consequences for the velocity distribution, but mainly for the transverse component of the bed shear stress that conditions the transverse bed slope.
3. The RANS computation did not yield the outer bank cell, where the LES computation did, apart from a region in the downstream area of the bend. This cell determines the near-bank hydrodynamics and is hence important as regards bank protection, bank erosion and river planimetry.
4. The RANS computation did not reproduce the internal shear layer at the inner bank, whereas the LES computation did, which is in a real river context relevant considering bank accretion, meander formation and river planimetry.
5. As a result of the four aforementioned issues, the RANS computation has resulted in very high bed shear stress, compared with the LES computations, mainly caused by the fact that high-momentum fluid adheres to the inner bank. Since the bed shear stresses are closely related to sediment transport processes, this has substantial consequences for the development of the river morphology.

Thirdly, what is the influence of the sub-grid closure model on the outcome of the LES in this respect? It was found that changes in the sub-grid model only yield minor differences in the outcome of the LES computation. This observation indicates that the essential role
is played by the large scales of turbulence, which interact with the mean secondary flow structures, whereas the small-scale turbulence is merely dissipative and does not play a dynamic role with respect to the time-averaged flow pattern.

Fourthly, what is the structure of the turbulence in curved open-channel flows? It was observed that the stream-wise-vertical Reynolds stresses clearly correlate with the rotational structure in the cross section, which has also been observed for straight open-channel flows. At the interface between the centre region cell and the outer bank cell at the free surface, the turbulence anisotropy is considerable, yielding low efficiency of the turbulent eddies in producing turbulence shear stresses.

Fifthly, what is the influence of the water depth on the flow characteristics in an open-channel bend? It was found that in the upstream part of the bend the strength of the centre region cell becomes stronger with increasing water depth. This is, however, not particularly the case in the downstream part of the bend. The outer bank cell was also found to gain strength with increasing water depth in the upstream part of the bend, whereas in the downstream part of the bend the outer bank cell loses strength in the experiment and even disappears in the LES computation. In none of the cases, the RANS computation showed this outer bank cell.

Acknowledgement

The numerical work is supported by the Dutch Technology Foundation (STW), Applied Science Division of NWO and the Technology Programme of the Ministry of Economic Affairs. The experimental work was supported by the Swiss National Science Foundation under grants 2100-052257.97 and 2000-059392.99.

References


