Simulation of open channel bend characteristics using computational fluid dynamics and artificial neural networks

Azadeh Gholami, Hossein Bonakdari*, Amir Hossein Zaji and Ali Akbar Akhtari

Department of Civil Engineering, Razi University, Kermanshah, Iran

(Received 22 November 2014; final version received 22 March 2015)

The ability to simulate flow characteristics is one of the most important issues in the design and application of open channel bends. Three-dimensional computational fluid dynamics (CFD) and multi-layer feed-forward artificial neural networks (MLFF-ANNs) are used and compared in modeling the flow depth and velocity field in sharp bends. CFD is modeled in two phases, water and air, using the volume of fluid method. The backpropagation algorithm is applied in the training process of the ANN model. An experimental study of a 90° curved channel is undertaken to verify and compare the efficiency of the CFD and ANN models. The results show that both CFD and ANN methods can be successfully applied to the modeling of open channel bend characteristics. However, the ANN method performs significantly better than CFD.

Keywords: artificial neural network; computational fluid dynamics; curved open channel; experimental modeling; velocity field

1. Introduction

Open channel bends are common hydraulic structures employed in natural and artificial channels. Because of the pressure gradient and centrifugal force, as well as the interaction between these two forces, strong secondary flows are created in open channel bends (Balen, Uijttewaal, & Blanckaert, 2010; Kimura, Uijttewaal, & Balen, 2008; Lien, Hsieh, Yang, & Yeh, 1999). Secondary flow causes full, three-dimensional complex fluid flow in bends (Ferguson, Parsons, Lane, & Hardy, 2003; Lu, Zhang, Cui, & Leung, 2004; Ottevanger, Blanckaert, & Uijttewaal, 2012).

Several experimental studies have been carried out on open channel bends to investigate flow characteristics. Rozovskii (1961) and Leschziner and Rodi (1979) studied sharp and mild bends, and developed some rules for specifying these. DeVriend and Geoldof (1983) conducted experimental studies on a 90° mild bend and reported that the change in the transverse profile of the water surface along the bend is linear. Experimental studies by Anwar (1986) demonstrated that the velocity pattern is not dependent on the radius at the bend’s entrance but the flow pattern’s bend-related effects are evident in the bend’s outlet. Ye and McCorquodale (1998) conducted a detailed, vast experimental study on curved channels. They examined the flow patterns and secondary currents in the bends and showed that secondary currents and super-elevation begin in the upstream cross sections and gradually reach the inner cross sections of the bend. Blanckaert and Graf (2001) conducted experimental studies on turbulent flow in a movable bed with a 120° sharp bend. Zeng, Constantinescu, Blanckaert, and Weber (2008) experimentally examined the velocity patterns, vorticity, turbulent kinetic energy, and bathymetry of a 193° sharp bend. Ramamurthy, Han, and Biron (2013) conducted vast experimental and numerical research studies on the velocity profiles of sharp bends. Many numerical studies have been conducted to model the characteristics of open channel bends as well.

One of the major benefits of numerical simulations of open channel bends is the transfer of discrete laboratory measurements to the continuous velocity field. DeMarchis and Napoli (2006) numerically modeled the flow pattern of a 270° sharp bend and examined the velocity components and water surface level within the channel. Zhang and Shen (2008) generated a three-dimensional numerical model of the flow within curved channels. The authors stated that the secondary flow caused by the centrifugal force is a primary reason for changes that occur in velocity and water surface level. Huang, Jia, Chan, and Wang (2009) used three-dimensional modeling along with different turbulence models to study secondary flow cells, depth average velocity distribution, and water surface level in curved channels. Furthermore, their model was able to simulate the minor secondary flow’s roll cell in the outer channel wall. Naji, Ghodsian, Vaghefi, and Panahpur (2010) used a numerical three-dimensional model to examine the flow pattern within a 90° bend and stated that

*Corresponding author. Email: bonakdari@yahoo.com
the secondary flow within this channel is the main reason why changes occur in longitudinal, transverse, and vertical velocity components. Ramamurthy et al. (2013) used two numerical models along with different turbulence models on a 90° sharp bend; they simulated the free surface using rigid lid, porosity, and volume of fluid methods. The authors stated that the numerical results of the RSM turbulence model and volume of fluid method are more consistent with the experimental results.

Soft computing methods have successfully been used in different areas of hydraulics (Bilhan, Emiroglu, & Kisi, 2010, 2011; Chau, Wu, & Li, 2005; Dursun, Kaya, & Firat, 2012; Ebtehaj & Bonakdari, 2013; Emiroglu, Bilhan, & Kisi, 2011; Jahangirzadeh et al., 2014; Kisi, Emiroglu, Bilhan, & Guven, 2012; Shamshirband et al., 2015; Wang, Chau, Cheng, & Qiu, 2009; Wang, Xu, Chau, & Lei, 2014; Wu & Chau, 2010; Wu, Chau, & Li, 2008; Yuhang and Wenxin, 2009). Sahu, Jana, Agarwal, and Khatua (2011) used artificial neural network (ANN) modeling to study and predict the velocity values within a meandering open channel. The high correlation coefficient between the results indicates the high accuracy of the presented network in predicting velocity within this channel. Bonakdari, Baghalian, Nazari, and Fazli (2011) predicted the velocity field values in a mild bend using ANN and the Genetic Algorithm. They used two-phase (air + water) three-dimensional modeling to verify the results where no experimental results were available. Their results indicated that there is an acceptable level of consistency between the results of the numerical and ANN models. Baghalian, Bonakdari, Nazari, and Fazli (2012) took a sediment tool into account and studied the flow in a 90° mild bend in a three-phase state (water + air + sediment). According to their results, the numerical and ANN models were more consistent with the experimental values compared with the analytical solution. Gholami et al. (2014) experimentally and numerically investigated the flow pattern in a 90° sharp bend. They focused on the general characteristics of flow and evaluated the performance of these models. It was found that sharp bends have a full, three-dimensional complex flow pattern.

According to the studies mentioned above, only a few have been conducted to evaluate the flow characteristics of open channel sharp bends using numerical methods. Therefore in this study, the computational fluid dynamics (CFD) and ANN methods are applied in order to model open channel sharp bend flow characteristics. An extensive experimental study is undertaken by the authors to verify and train the models. The CFD models were run using FLUENT software with the $k-\varepsilon$ (RNG) turbulence method (Fluent Manual, 2005). Two different ANN models were developed to model the velocity field and flow depth. After comparing the performance of the CFD and ANN models, their performance was compared in case there are no experimental measurements available. The present study results regarding sharp bends are very efficient in the design and construction of artificial curved channels.

2. Experimental model

The ANN models were trained and tested and the CFD models validated using an experimental study conducted in the hydraulic laboratory at the Ferdowsi University of Iran (Figure 1). A square Plexiglas flume of 40.3 cm width and height was used. The set-up consisted of a straight channel with a 3.6 m entrance, a sharp open channel bend ($R_c/b = 1.5 < 3$) with a central angle of 90°, followed by a straight channel of 1.8 m length.

To measure the longitudinal velocity, the PROPLER one-dimensional velocity meter was used. The flow height was measured with a micrometer. During the experiments, flow discharge, downstream depth, Froude number, and Reynolds number were fixed at 25.3 lit/s, 15 cm, 0.34, and 44,705, respectively. Therefore, the flow was considered fully turbulent and subcritical. A more detailed explanation of the experimental study conducted is given in Gholami et al. (2014).

To perform this study, the velocity and depth at different points and sections in the channel were measured. Considering the effects of the channel walls and floor on the velocity index, 13 transverse points were selected in the channel width; at each transverse point, four depth points...
were selected at 3, 6, 9 and 12 cm from the water surface. Figure 2 indicates the coordinates of the points used for measuring the velocity in each cross section. The positions of these sections are 40 cm before the bend, 40 and 80 cm after the bend, the beginning of the bend, the end of the bend, and the sections at the central angles 22.5°, 45°, and 67.5° in the bend (Figure 3).

3. Computational fluid dynamics (CFD) model

3.1. Governing equations

Governing equations for viscous and turbulent noncompressible flows include the continuity equation (Equation 1) and Reynolds-Averaged Navier-Stokes equation (Equation 2), written as follows:

\[ \frac{\partial U_i}{\partial x_i} = 0, \quad \text{with} \quad i = 1, 2, 3, \quad (1) \]

\[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} - \rho \delta_{ij} \frac{\partial U_i}{\partial x_j} \quad (i,j = 1, 2, 3), \quad (2) \]

where \( U_i \) is the average velocity component in the \( x_i \) direction, \( \rho \) is the density of the water, \( P \) is the pressure, \( \delta_{ij} \) is the Kroncker delta, which is 1 for \( i = j \) and 0 otherwise, and \( x_i \) and \( x_j \) are the general space dimensions. The Reynolds stress term is modeled by the approximation (Stoesser, Ruether, & Olsen, 2010):

\[ \overline{u_i u_j} = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \left[ \frac{2}{3} k \delta_{ij} \right]. \quad (3) \]

In the case of the \( k - \varepsilon \) (RNG) model, the turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \) for determining the eddy viscosity are obtained with the following equations (Rahimzadeh, Maghsoodi, Sarkardeh, & Tavakkol, 2012):

\[ \nu_t = C_{\mu} \frac{k^2}{\varepsilon}, \quad (4) \]

\[ \frac{\partial (k)}{\partial t} + \frac{\partial (k U_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \varepsilon, \quad (5) \]

\[ \frac{\partial (\varepsilon)}{\partial t} + \frac{\partial (\varepsilon U_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \alpha_{\varepsilon} \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right] - C_{1_\varepsilon} \varepsilon \frac{G_k + C_{3_\varepsilon} G_b}{k} - \frac{C_{2_\varepsilon} \varepsilon^2}{1 + \beta \eta^3} \frac{\varepsilon^2}{k}, \quad (6) \]

where \( \mu_{eff} \) is the effective viscosity, \( G_k \) and \( G_b \) are generation of turbulence kinetic energy due to the mean velocity gradients and generation of turbulence kinetic energy due to buoyancy, respectively, \( \alpha_k \) and \( \alpha_{\varepsilon} \) are the inverse Prandtl numbers for \( k \) and \( \varepsilon \), respectively, and \( \alpha_k \) and \( \alpha_{\varepsilon} \) are computed using the following formula:

\[ \left[ \frac{\alpha - 1.3929}{\alpha_0 - 1.3929} \right]^{0.6321} = \left[ \frac{\alpha + 2.3929}{\alpha_0 + 2.3929} \right]^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}}, \quad (7) \]

where \( \alpha_0 = 1.0 \). In the high Reynolds number limit (\( \mu_{mol} / \mu_{eff} \ll 1 \)), \( \alpha_k = \alpha_{\varepsilon} \approx 1.393 \).

In the present study, the parameters \( C_{1_\varepsilon}, C_{2_\varepsilon}, \) and \( C_{3_\varepsilon} \) are constant and given the standard values of 0.0845, 1.42, and 1.68, respectively.

3.2. Gridding, CFD solving methods and boundary conditions

Gridding was constructed using GAMBIT, which is the preprocessor available in the commercial software FLUENT package. The execution time of computations from CFD modeling and the resulting accuracy and degree of consistency with the experimental results depend on the gridding size. A number of different grids of various sizes were examined in order to model this experimental study and to consider the initial results obtained. After carrying out sensitivity analysis on the meshing, a grid was eventually selected with a total of 67,500 nodes (25, 30, and 90 nodes in the length, width, and height, respectively) for a 90° bend. In order to regulate the solution field gridding near the bending, the smaller grid of the walls and water’s free surface was determined suitable due to the steeper gradient. In addition, the grid was selected in such manner that it would be in line with the flow direction. As it approaches the bend the grid becomes smaller, while as it exits the bend the grid becomes larger with the aim to save computation time. Figure 4 illustrates a view of the solution field in the plan and cross section.

FLUENT software was used for CFD modeling in this research. This software uses the finite volume method.
(FVM) to solve the three-dimensional equations which govern flow. It considers the entire flow field as discrete control volumes and calculates the integral of the equations governing fluid flow on each control volume. The software uses different discretization schemes determined by the user, solves algebraic discrete equations and uses different algorithms to solve dependent equations to the point of reaching convergence.

In this study, the PISO pressure-velocity coupling algorithm was used because it is designed specifically for transient simulations. The PRESTO discretization scheme was used for pressure. The modified High Resolution Interface Capturing (HRIC) for volume fraction, QUICK, was applied for momentum and first-order upwind for turbulent kinetic energy discretization schemes (Fluent Manual, 2005). The flow was analyzed three-dimensionally and VOF served to model the two-phase flow of the free surface and solve the flow field until the remainder reached $10^{-4}$. The boundary condition was also applied to the model in order to determine the solution shown in Table 1.

---

**Figure 4.** Gridding in CFD Model for (a) plan, and (b) cross section.

**Table 1.** Boundary conditions applied in the CFD model.

| Boundary conditions | Type of boundary condition | Notes | Values |
|---------------------|---------------------------|-------|--------|
| Inlet               | Velocity inlet            | The inlet was considered separately for air and water phase states. | $V_{\text{(water)}} = 0.419 \text{ m/s}$ $V_{\text{(air)}} = 0.0001 \text{ m/s}$ |
| Free surface        | Pressure inlet            | Two-phase flow. In open channel and free surface level states. Atmospheric pressure | – |
| Outlet              | Pressure outlet           | – | – |
| Bed and walls       | Wall                      | Using the standard wall function. | – |
4. Application of artificial neural network (ANN) models

4.1. ANN structure

The ANN is an idea inspired by biological neural networks and is used to process data. The MATLAB R2011b software was utilized in order to prepare a proper ANN model (MATLAB, 2009). Two separate neural networks were used in this study: one to predict velocity (case 1) and the other to predict water depth (case 2). Both cases are multi-layer feed-forward (MLFF) neural networks. Figure 5 provides a general view of these networks, which are organized into three layers, namely input, hidden, and output (there may be one or more hidden layers). Each layer is made up of a number of neurons which are connected to the neurons of the adjacent layer through some weights. The network training process entails correcting the connective weights between neurons through the best training method.

In this study, 3, 40, and 1 neurons were used in the input layer, one hidden layer, and output layer, respectively, in case (1). The input for this case is the coordinate of points in the $X$, $Y$, and $Z$ directions at various points in the channel and the output or target is the corresponding velocity of these points.

2, 10, 10, and 1 neurons were selected to be placed in the input layer, two hidden layers, and the output layer, respectively, in case (2). In this case, the input is the coordinate of the points in the $X$ and $Y$ directions at various channel points, while the output or target is the corresponding flow depth of each of these points.

Careful and actual analyses for the purpose of finding the number of neurons in the hidden layer are generally very complex. There is no particular law which specifies the number of neurons within hidden layers (Dawson & Wilby, 1998), but we can say that the number of hidden layer neurons is a function of the number of input axial elements and also the maximum number of input-space regions which can be linearly separated (Fausett, 1994; Lippman, 1987). Therefore, the number of neurons in the hidden layer is usually obtained experimentally. If the number of neurons is excessively low it reduces the analysis capability and consequently the numerical accuracy of prediction. However, if the number of hidden layer neurons is excessively high, the model will undergo over-training and memorize the data instead of analyzing them. The number of neurons within the hidden layer(s) is commonly determined through trial and error (Cobaner, Unal, & Kisi, 2009; Kalteh, 2008; Kisi, 2008; Zaji & Bonakdari, 2015). Therefore, the trial and error method was utilized and different models were tested with various numbers of neurons considered within the hidden layer. Then the model that presented the best results was selected as the final ANN model. The training process continued for 1000 iterations, until both networks reached an acceptable level of error between the data obtained from the ANN model and the experimental data. Both networks were trained through the error backpropagation method in this research, and the tansig function served to activate the neurons in the hidden and output layers. This equation is shown below:

$$f(x) = \frac{2}{1 + \exp(-2x)} - 1.$$

A total of 416 experimental data were used in the present study. For case 1 the data were divided into two groups: 320 data for training the network and 96 for testing. 104 experimental data were used for case 2, with 80 data used to train the network and 24 to test it. To train both neural networks, the experimental data of flow velocity and depth of flow values at 13 different points of the channel width (see Figure 2) in the eight cross sections (40 cm before the bend, $0^\circ$, $22.5^\circ$, $45^\circ$, $67.5^\circ$, and $90^\circ$, and 40 and 80 cm after the bend) were used (see Figure 3). In case 2, however, the depth corresponding to these points in the channel bed was used and the velocity corresponding to these points at four different distances from the channel bed ($Z = 3, 6, 9, \text{and } 12 \text{ cm}$) was applied in case 1.

4.2. The error backpropagation method

In order to train the network with the error backpropagation algorithm, the inputs are presented to the neural network.
and the outputs obtained from the network are compared with the actual output values (target values) in order to compute the error. Then the computed error within the network is propagated backwards and the connected weights are corrected. This process continues until an acceptable level of convergence is achieved (Kalteh, 2008).

The weight changes in each $W_{ji}$ specific weight, which connects the $j$th neuron to the $i$th neuron in the previous layer. It can be written as follows, where $\eta$ is the training rate parameter and $E$ is the error function (Rajurkar, Kothyari, & Chaube, 2004):

$$W_{ji} = -\eta \frac{\partial E}{\partial W_{ji}}. \tag{9}$$

The weight changes can be written as shown below using the chain rule in partial derivatives (Rajurkar et al., 2004):

$$\Delta W_{ji}^n = \eta^n \delta^n_j z^n_{j-1}. \tag{10}$$

where $z^n_{j-1}$ indicates the output value of the $j$th neuron in the $(n-1)$th layer, $\delta^n_j = (T_i - z^n_j)g'(v^n_j)$ is for the output layer, $\delta^n_j = (\sum_{h=1}^n \delta^{n+1}_h W^n_{bh})g'(v^n_j)$ is for the hidden layers, $v_j = W^n_{bh} u^n_h$, and $g()$ indicates the first derivative of the nonlinear activation function (Rajurkar et al., 2004).

### 4.3. Data normalizing and model performance evaluation

All input data are normalized as data in the range of 0 to 1 in the ANN model. Equation (11) is applied in the present study to normalize the data (Mohd Zahit, 2007):

$$X_n = \frac{X_i - X_{\min}}{X_{\max} - X_{\min}}, \tag{11}$$

where $X_n$ is the normalized parameter, $X_{\max}$ is the maximum observational parameter, $X_{\min}$ is the minimum observational parameter and $X_i$ is the observational parameter. It is crucial to determine the indexes and issues in order to analyze the results obtained from the models in the simulation results.

The performance of both the CFD model and the ANN model was examined through the Root Mean Square Error (RMSE), Mean Absolute Error (MAE) and coefficient of determination $R^2$ statistical parameters in the present research:

Equations (12) and (13) are for computing $RMSE$ and $MAE$, respectively:

$$RMSE = \sqrt{\frac{\sum (O_i - t_i)^2}{N}}, \tag{12}$$

$$MAE = \frac{1}{N} \sum_{i=1}^{N} |O_i - t_i|. \tag{13}$$

Equation (14) is for calculating $R^2$:

$$R^2 = 1 - \frac{\sum (O_i - t_i)^2}{\sum (O_i - \bar{O}_i)^2}. \tag{14}$$

In the above-mentioned equations, $O_i$ is the input observational parameter, $t_i$ is the parameter predicted by the ANN model, $\bar{O}_i$ is the mean observational parameter and $N$ is the number of parameters. $R^2$ is the linear regression line between the values predicted by the ANN model and the observational values to determine the network application.

Figure 6. Distribution of longitudinal component of velocity at different sections and levels: (a) at water level $Z = 3$ cm (near the channel bed), and (b) at water level $Z = 12$ cm (near the water surface).
5. Results and discussion

5.1. The CFD model

5.1.1. CFD model performance evaluation

The experimental results were used to examine the accuracy of the CFD model. Figure 6 compares the longitudinal velocity values within different cross sections at two levels, \( Z = 12 \text{ cm} \) (near the water surface) and \( Z = 3 \text{ cm} \) (near the channel bed), which were obtained from the CFD model and the experimental values. The predicted transverse water surface profiles of the CFD model were compared with the experimental measurements at different cross sections in Figure 7. The CFD model performance evaluation for velocity \( (Z = 3 \text{ and } 12 \text{ cm}) \) and flow depth prediction is presented in Table 2.

Clearly, the consistency of the CFD and experimental results is generally acceptable, with an average RMSE and MAE of 2.05 and 1.8, respectively, for the level near the channel bed, and 2.07 and 1.52 for the levels near the water surface in velocity prediction. Regarding the velocity prediction values in Table 2, it can be stated that when \( Z = 3 \text{ cm} \) (near the channel bed) the minimum error occurs in the cross section located 40 cm after the bend and the maximum error occurs near the middle cross section of the bend. In contrast, when \( Z = 12 \text{ cm} \) (near the water surface) the minimum error value occurs in the middle cross section and the maximum error occurs in the cross section located 40 cm after the bend. This is because velocity decreases (due to secondary flows) beginning from the layers close to the water surface \( (Z = 12 \text{ cm}) \) until it reaches the lower layers \( (Z = 3 \text{ cm}) \) at the end cross sections of the bend. Basically, when velocity decreases in the upper layers it does not decrease in the layers close to the bed, subsequently creating an insignificant difference between the error values obtained from the CFD model and the results of the experimental model in the cross sections located after the bend and in the layers close to the water surface.

With regard to the depth prediction values in Table 2, CFD predicts flow depth with an RMSE and MAE of 0.13 and 0.11, respectively. In addition, it can be concluded that the amount of prediction error increases by moving from the beginning toward the end sections of the bend. This is due to the greater impact of the bend on fluid flow in the end sections.

As the flow enters the bend, the transverse pressure gradient between the inner and outer wall causes the velocity to reach a maximum in the inner wall and minimum in the outer wall. The velocity increase continues up to the 67.5° cross section and the maximum velocity continues in this area. The development of secondary flows in the final cross sections of the bend causes the maximum velocity to gradually move to the channel axis from the inner wall and then move towards the outer channel wall. Figure 8 shows the geometrical locations of the maximum velocity at different cross sections in the channel in the 3, 6, 9, and 12 cm layers from the channel bed. A careful study of this figure reveals that the maximum velocity occurs in the inner channel at half width and in different layers from the channel bed to the end of the bend because the bend is sharp. The maximum velocity is transferred to the outer wall in the cross sections after the bend. Molls and Chaudhry (1995) and Leschziner and Rodi (1979) found similar results from conducting studies on sharp bends. The velocity values in different channel layers will be explained thoroughly in the following sections as well as studying the longitudinal velocity-depth profile distribution.

5.1.2. Secondary currents

Secondary flows are obvious flow phenomena within curved channels. Figure 9 indicates the transverse velocities and secondary flow cells at 0°, 45°, and 90°, and 80 cm after the bend cross section. A one-way radial
Table 2. Comparison of the velocity distribution and depth of flow prediction in the CFD model and the experimental model at \( Z = 3 \) and \( Z = 12 \) cm from the bed at different cross sections.

| Cross section       | Velocity prediction | Depth prediction |
|---------------------|---------------------|-----------------|
|                     | RMSE (m) MAE (m)    | RMSE (m) MAE (m)| RMSE (m) MAE (m) |
|                     | \( Z = 3 \) cm      | \( Z = 12 \) cm | \( Z = 0 \)     |
| 40 cm before the bend| 2.58 2.3           | 1.66 1.53      | 0.048 0.046     |
| 0°                   | 2.85 2.43          | 1.65 1.29      | 0.11 0.098      |
| 22.5°                | 3.03 2.7           | 1.62 1.45      | 0.096 0.087     |
| 45°                  | 2.66 2.47          | 1.06 0.918     | 0.12 0.096      |
| 67.5°                | 2.18 1.91          | 2.45 1.33      | 0.22 0.201      |
| 90°                  | 1.03 0.96          | 2.41 1.56      | 0.15 0.105      |
| 40 cm after the bend | 0.94 0.77          | 1.93 1.34      | 0.13 0.12       |
| 80 cm after the bend | 1.15 0.93          | 3.8 2.76       | 0.165 0.164     |
| Averaged values      | 2.05 1.8           | 2.07 1.52      | 0.13 0.11       |

Figure 8. The geometrical location of the maximum velocity in different layers of the channel bed.

flow occurs in the initial cross section of the bend, to establish the continuity of the flow from the outer to the inner wall. In the following cross sections, the main secondary flow cell can be seen in all cross-sectional widths, and as it moves forward along the bend it can be seen moving towards the inner channel wall. The minor secondary flow cells are also clearly visible in the outer wall of the channel near the water surface. Blanckaert and Graf (2001) conducted experimental studies on 120° sharp bends while Huang et al. (2009) used a numerical model and also reported the existence of minor secondary flow cells. The minor roll cell indicates the \( k - \varepsilon \) (RNG) turbulence model’s capability to predict the cell, since in case of using the \( k - \varepsilon \) (standard) isotropic (linear) turbulence model this cell is not predicted (Naji et al., 2010).

5.2. The ANN models

As previously mentioned, the defined neural networks are first trained using existing experimental data and the error backpropagation method, after which they are tested. The general specifications of both networks utilized are presented in Table 3. The correlation analysis between the data predicted by the ANN model and the experimental data is presented in Figure 10 for case 1 and Figure 11 for case 2. The \( R^2 \) values are shown for each figure, where \( R^2 \) is equal to 0.9834 for case 1 and 0.9801 for case 2 in the training state.

The performance of the ANN models’ velocity and depth predictions is shown in Table 4, where the velocity fields are investigated at two heights, \( Z = 3 \) and 12 cm. According to this table, the ANN model that predicted velocity with an RMSE of 0.59 and 0.52 for \( Z = 3 \) and 12 cm, respectively, and flow depth with an RMSE of 0.037, performed fairly well in open channel bend characteristics prediction. Also according to Table 4, at \( Z = 3 \) cm, the velocity prediction performance was the highest in the cross sections located before and at the entrance of the bend, but at \( Z = 12 \) cm, the velocity prediction performance was highest at the cross section located after the bend. Moreover, the ANN model did not adequately predict the velocity decrease from the layers close to the water surface in the section located after the bend. This result is consistent with the result obtained in Section 5.1.1. Thus, it can be stated that the error value of both models (ANN and CFD) slightly increased when predicting the effects of secondary flows on the bend’s flow patterns. In case 2 (water surface prediction model), the minimum error occurred at the entrance cross sections of the bend and the maximum error value at the end cross sections of the bend. It can be concluded that this model did not perform well in predicting flow depth, as it became constant in the cross sections located after the bend (returning to the primary normal depth of 15 cm) and so it slightly differed from the experimental values. Therefore, similar to the CFD model, the ANN was less accurate in the middle and end sections, and the section located after the bend where the impact of the bend is greater.

Considering these values and also Figures 10 and 11, it can be understood that both networks were trained well
and their performance in predicting velocity and flow depth parameters was satisfactory. Therefore, after ensuring performance accuracy of these networks, they can be used to predict the velocity and flow depth values at channel points where there are no available experimental data, as shown in the following section.

5.3. Comparison of the CFD and ANN model results

As mentioned earlier, the trained neural networks are used in this section to examine the water surface profile and velocity distribution at points in the channel where no experimental data are available. These values are finally compared with the values obtained from the CFD simulation. Experimental data were obtained at only eight cross sections in the present research, but ANN models can help to examine and study the water depth and velocity values at other cross sections.

5.3.1. Longitudinal velocity distribution in the channel width and depth

The transverse distributions of the depth-averaged longitudinal velocity at 20°, 40°, 50°, 60°, and 70°, and 30 cm after the bend cross sections as simulated by the CFD and ANN models (case 1) are compared in Figure 12. The RMSE and MAE values for the results of the CFD and ANN model results are shown in Table 5. In terms of the average values of the RMSE (2.33) and MAE (2.02), both methods were able to predict the depth-averaged velocity at different cross sections very well. There is also an acceptable level of consistency between the results of both models. The geometrical location of the maximum depth-averaged velocity at different cross sections also supports this in Figure 13. Considering this figure and as mentioned in previous sections, the maximum velocity commonly occurs in the inner section of the bend, the reason being the power of the longitudinal flow and the fact that it overpowers the secondary flows within these types of bends. This result is also consistent with the results obtained by Zhou, Wang, Shao, and Jia (2006). When examining the velocity pattern in the present 90° sharp bend and taking into account Figure 12, the crucial point is that the transverse distribution of the depth-averaged velocity is linear from the beginning to almost the end of the bend, where the secondary flow becomes stronger.

Table 3. Overview of the two ANN models.

| Properties                  | Case 1 | Case 2 |
|-----------------------------|--------|--------|
| Number of data              | 416    | 104    |
| Number of training data     | 320    | 80     |
| Number of testing data      | 96     | 24     |
| Input variables             | X, Y   | X, Y   |
| Output variables            | Velocity, Water depth |
| Number of hidden layers     | 1      | 2      |
| Input neurons               | 3      | 2      |
| Hidden 1 neurons            | 40     | 10     |
| Hidden 2 neurons            | 1      | 1      |
| Output neurons              | 1      | 1      |
| RMSE of training data       | 0.57   | 0.039  |
| RMSE of testing data        | 0.55   | 0.037  |
| MAE of training data        | 0.48   | 0.05   |
| MAE of testing data         | 0.45   | 0.03   |
| R² of training data         | 0.9834 | 0.9801 |
| R² of testing data          | 0.9844 | 0.9922 |

Figure 9. Secondary currents at (a) 0°, (b) 45°, (c) 90°, and (d) 80 cm after the bend.
In Figure 14, the depth distributions of the longitudinal velocity component at 20°, 50°, and 70°, and 30 cm after the bend cross sections are compared through the CFD and ANN models.

The acceptable level of consistency between the results of the two models can clearly be seen in these figures. In Figure 14, for the cross section located after the bend, the ANN model predicted lower velocity values between the water surface and channel mid-depth, which is slightly different from the CFD model results. In general, both the CFD model and the ANN model are capable of predicting the velocity pattern in different channel cross sections. The results obtained from both models with a mean RMSE of 2.5 are acceptably consistent with each other. The velocity

Table 4. Comparison of the velocity and depth of flow prediction in the ANN model and the experimental model at \(Z = 3\) and \(Z = 12\) cm from the bed at different cross sections.

| Cross section       | Velocity prediction | Depth prediction |
|---------------------|---------------------|-----------------|
|                     | RMSE (m)    MAE (m) | RMSE (m)    MAE (m) | RMSE (m) | MAE (m) |
| 40 cm before the bend | 0.79 0.57 | 0.47 0.39 | 0.021 | 0.02 |
| 0°                  | 0.75 0.57 | 0.5 0.42 | 0.036 | 0.03 |
| 22.5°               | 0.6 0.54  | 0.48 0.41 | 0.0265 | 0.017 |
| 45°                 | 0.54 0.44  | 0.47 0.4 | 0.023 | 0.02 |
| 67.5°               | 0.54 0.3 | 0.52 0.42 | 0.058 | 0.049 |
| 90°                 | 0.53 0.49 | 0.53 0.49 | 0.039 | 0.03 |
| 40 cm after the bend | 0.55 0.45 | 0.56 0.52 | 0.059 | 0.049 |
| 80 cm after the bend | 0.42 0.38 | 0.65 0.57 | 0.038 | 0.028 |
| Averaged values     | 0.59 0.46 | 0.52 0.45 | 0.037 | 0.03 |
depth distribution is not logarithmic in any of the states. Ghamry and Steffler (2003) suggested that the depth velocity distribution is parabolic for the flow within channels. The results obtained from the CFD and ANN models are slightly different from each other in cross sections where the presence of the bend and its main characteristic (secondary flow) in the channel lead to changing flow patterns and velocity component distributions, as mentioned in prior sections.

5.3.2. Water surface profile

The longitudinal profile of the water surface predicted by the CFD model and the ANN model is illustrated in Figure 15 along the channel length, 30 cm before and 30 cm after the bend, as well as near the inner walls, outer walls, and the channel axis. The water surface rises in the cross section before the bend in both models, since the water needs to gain more power to enter the bend. As it exits the bend, the flow depth is restored to its original state (15 cm). Considering the figure and the mean RMSE of 1.3, both models were able to predict the water surface longitudinal profile well and both are acceptably consistent with each other except in the inner edges of the bend, as mentioned before. With respect to examining the flow pattern within sharp bend channels, Ye and McCorquodale (1998) also point out that water surface rises in the cross sections located before the bend.

Table 6 shows the water surface difference in the outer mid-width of the channel (subtraction of the water surface in the channel axis from the outer wall) and the inner mid-width (subtraction of the water surface in the inner wall from the channel axis) in the initial, end, and inner cross sections of the 90° bend with the CFD and ANN models. The values in this table signify that in all cross sections,
both models predicted significantly greater water surface change in the inner mid-width than the outer mid-width of the channel. Therefore, it can be stated that the transverse slope of the water surface is nonlinear in the 90° sharp bend. With regard to examining the flow pattern in sharp bends, Leschziner and Rodi (1979) concurred that the transverse slope of the water surface is nonlinear in sharp bends and the inner half has a greater slope than the outer half of the channel. In addition, Bodnar and Prihoda (2006) conducted a series of numerical studies on
Table 6. The water surface difference in the outer and inner mid-width of the channel cross sections in the CFD and ANN models.

| Mid-width | 0° (entrance) | 20° | 40° | 60° | 80° | 90° (end of the bend) |
|-----------|---------------|-----|-----|-----|-----|-----------------------|
|           | Inner | Outer | Inner | Outer | Inner | Outer | Inner | Outer | Inner | Outer | Inner | Outer |
| CFD | 0.31 | 0.16 | 0.7 | 0.33 | 0.59 | 0.46 | 0.63 | 0.45 | 0.58 | 0.44 | 0.22 | 0.19 |
| ANN | 0.42 | 0.21 | 0.83 | 0.26 | 0.64 | 0.14 | 0.89 | 0.37 | 0.87 | 0.48 | 0.35 | 0.28 |

Figure 15. Comparison of longitudinal water surface profiles with the CFD model and the ANN model at the outer edge of the bend, channel axis, and inner edge of the bend.

In light of the above discussions, it can be concluded that both the CFD model and the ANN model are capable of predicting the general pattern governing flow, and the results obtained from both models are consistent. Therefore both methods can be used for the purpose of predicting flow pattern within bends. Nonetheless, each method has advantages and disadvantages.

The CFD model is more time-consuming than the ANN model. The ANN model helps achieve a desirable network in a shorter time for predicting values through network training. In addition, the ANN model can predict flow variable values at any point without requiring the CFD model or available experimental data.

6. Conclusion

The CFD and ANN methods were applied in this study to simulate flow characteristics in open channel bends. The CFD and ANN models were verified and trained through an experimental study. Two ANN models were developed, one for modeling the velocity field and another for modeling the flow depth. After that, the performance of the CFD and ANN models in velocity and flow depth simulation was compared. The results show that both models are reasonably accurate. However, the ANN models with an RMSE of 0.55 and 0.037 in velocity and depth prediction, respectively, performed significantly better than the CFD model with an RMSE of 2.06 and 0.13. In the next part of the study, the application of validated CFD and ANN models were investigated for cross sections where experimental data is not available. The results indicate that both the CFD and ANN models can predict flow characteristics without experimental data with an RMSE of 2.33.

Notations

- $R_c$: Central radius of bend
- $b$: Channel width (m)
- $U_i$: Average velocity component (m/s)
- $\rho$: Density of water (kg/m³)
- $p$: Pressure
- $\delta_{ij}$: Kronecker delta
- $x_i$: The general space dimension in the $x$ direction
- $x_j$: The general space dimension in the $y$ direction
- $-\rho u_i'u_j$: Reynolds stresses
- $\bar{u}_i$: Mean velocity component
- $u'_i$: Fluctuating velocity component
- $\mu$: Viscosity of fluid
- $\mu_\tau$: Turbulent viscosity
- $\mu_{\text{eff}}$: Effective viscosity
- $\mu_{\text{mol}}$: Molecular viscosity
- $t$: Time (s)
- $\beta$: Constant number
- $\varepsilon$: Turbulence dissipation rate (m²/s³)
- $k$: Turbulence kinetic energy (m²/s³)
- $G_k$: Generation of turbulence kinetic energy due to the mean velocity gradients
- $G_b$: Generation of turbulence kinetic energy due to buoyancy
- $S$: Modulus of the mean rate-of-strain tensor
- $\alpha_k$: Inverse effective Prandtl number for $k$
- $\alpha_\varepsilon$: Inverse effective Prandtl number for $\varepsilon$
- $C_{1k}$: $k - \varepsilon$ Turbulence model constant
- $C_{1\varepsilon}$: $k - \varepsilon$ Turbulence model constant
- $C_{2k}$: $k - \varepsilon$ Turbulence model constant
- $W_{ij}$: Specific weight which connects the $j$th neuron to the $i$th neuron in the previous layer of the ANN
- $\eta$: Learning rate parameter of the ANN
- $E$: The error function of the ANN
- $W^e_{hj}$: Weight between the hidden layer and the output layer of the ANN
ui
Input of the ANN

un
Output of the hidden layer of the ANN

δn
The local gradient function in the nth output layer of the ANN

δh
The local gradient function in the nth hidden layer of the ANN

g(θ)
The first derivative of the nonlinear activation function of the ANN

Ti
Desired output of the ANN

Disclosure statement
No potential conflict of interest was reported by the authors.

References
Anwar, H. O. (1986). Turbulent structure in a river bed. Journal of Hydraulic Engineering, 112(8), 657–669.
Baghalian, S., Bonakdari, H., Nazari, F., & Fazli, M. (2012). Closed-form solution for flow field in curved channels in comparison with experimental and numerical analyses and artificial neural network. Engineering Applications of Computational Fluid Mechanics, 6(4), 514–526.
Balen, W. V., Uijttewaal, W. S. J., & Blanckaert, K. (2010). Large-eddy simulation of a curved open-channel flow over topography. Physics of Fluids, 22(7), 1–18.
Bilhan, O., Emiroglu, M. E., & Kisi, O. (2010). Application of two different neural network techniques to lateral outflow over rectangular side weirs located on a straight channel. Advances in Engineering Software, 41(6), 831–837.
Bilhan, O., Emiroglu, M. E., & Kisi, O. (2011). Use of artificial neural networks for prediction of discharge coefficient of triangular labyrinth side weir in curved channels. Advances in Engineering Software, 42(4), 208–214.
Blanckaert, K., & Graf, W. H. (2001). Mean flow and turbulence in open channel bend. Journal of Hydraulic Engineering, 127(10), 835–847.
Bodnar, T., & Prihoda, J. (2006). Numerical simulation of turbulent free-surface flow in curved channel. Flow, Turbulence and Combustion, 76, 429–442.
Bonakdari, H., Baghalian, S., Nazari, F., & Fazli, M. (2011). Numerical analysis and prediction of the velocity field in curved open channel using artificial neural network and genetic algorithm. Engineering Applications of Computational Fluid Mechanics, 5(3), 384–396.
Chau, K. W., Wu, C. L., & Li, Y. S. (2005). Comparison of several flood forecasting models in Yangtze river. Journal of Hydrologic Engineering, 10(6), 485–491.
Cobaner, M., Unal, B., & Kisi, O. (2009). Suspended sediment concentration estimation by an adaptive neuro-fuzzy and neural network approaches using hydro-meteorological data. Journal of Hydrology, 367(1), 52–61.
Dawson, C. W., & Wilby, R. L. (1998). An artificial neural network approach to rainfall-runoff modeling. Hydrological Sciences Journal, 43(1), 47–66.
DeMarchis, M., & Napoli, E. (2006). 3D numerical simulation of curved open channel flows. Proceedings of the International Conference on Water Resources, Hydraulics & Hydrology, May 11–13, Chalkida, Evia Island, Greece, 86–91.
DeVriend, H. J., & Geolof, H. J. (1983). Main flow velocity in short river bends. Journal of Hydraulic Engineering, 109(7), 991–1011.
Dursun, O. F., Kaya, N., & Firat, M. (2012). Estimating discharge coefficient of semi-elliptical side weir using ANFIS. Journal of Hydrology, 426, 55–62.
Ebtehaj, I., & Bonakdari, H. (2013). Evaluation of sediment transport in sewer using artificial neural network. Engineering Applications of Computational Fluid Mechanics, 7(3), 382–392.
Emiroglu, M. E., Bilhan, O., & Kisi, O. (2011). Neural network for estimation of discharge capacity of triangular labyrinth side-weir located on a straight channel. Expert Systems with Applications, 38(1), 867–874.
Fausett, L. (1994). Fundamental of neural networks: Architecture, algorithms and applications. New Jersey, USA: Prentice Hall.
Ferguson, R. I., Parsons, D. R., Lane, S. N., & Hardy, R. J. (2003). Flow in meander bends with recirculation at the inner bank. Journal of Water Resources Research, 39(11), 1–13.
Fluent Manual (2005). Manual and user guide of fluent software. New York, USA: Fluent Inc.
Ghamry, H. K., & Steffler, P. M. (2003). Effect of applying different distribution shapes for velocities and pressure on simulation of curved open channels. Journal of Hydraulic Engineering, 128(11), 969–982.
Gholami, A., Akhtari, A. A., Minatour, Y., Bonakdari, H., & Javadi, A. A. (2014). Experimental and numerical study on velocity fields and water surface profile in a strongly-curved 90° open channel bend. Engineering Applications of Computational Fluid Mechanics, 8(3), 447–461.
Huang, S. L., Jia, Y. F., Chan, H. C., & Wang, S. S. Y. (2009). Three dimensional numerical modeling of secondary flows in a wide curved channel. Journal of Hydrodynamics, 21(6), 758–766.
Jahangerzadeh, A., Shamshirband, S., Petkovic, D., Basser, H., Sedaghat, A., Akib, S., & Karami, H. (2014). Adaptive neuro-fuzzy estimation of the influence of slot on local scour at bridge pier groups. Journal of Coastal Conservation. doi:10.1007/s11852-014-0357-5.
Kalteh, A. M. (2008). Rainfall-runoff modeling using artificial neural networks (ANNs): modeling and understanding. Caspian Journal of Environmental Science, 6(1), 53–58.
Kimura, I., Uijttewaal, W. S. J., & Balen, W. (2008). Application of the non-linear k – ε model for simulation curved open channel flows. Proceedings of River Flow, September 3–5, Cesme-izmir, Turkey, 1, 99–108.
Kisi, O. (2008). The potential of different ANN techniques in evapotranspiration modeling. Hydrological Processes, 22(14), 2449–2460.
Kisi, O., Emiroglu, M. E., Bilhan, O., & Guven, A. (2012). Prediction of lateral outflow over triangular labyrinth side weirs under subcritical conditions using soft computing approaches. Expert Systems with Applications, 39(3), 3454–3460.
Leschziner, M. A., & Rodi, W. (1979). Calculation of strongly curved open channel flow. Journal of the Hydraulics Division, 105(10), 1297–1314.
Lien, H. C., Hsieh, T. Y., Yang, J. C., & Yeh, K. C. (1999). Bend flow simulation using 2D depth-averaged model. Journal of Hydraulic Engineering, 125(10), 1097–1108.
Lippman, R. P. (1987). An introduction to computing with neural nets. ASSP Magazine IEEE, 4(2), 4–22.
Lu, W. Z., Zhang, W. S., Cui, C. Z., & Leung, A. Y. T. (2004). A numerical analysis of free-surface flow in curved open channel with velocity-pressure-free-surface correction. Computational Mechanics, 33, 215–224.
MATLAB (2009). User’s Guide Version 7.8, R2009a. MathWorks Co., USA.
Mohd Zahit, A. (2007). The application of the artificial neural network model for river water quality classification with emphasis on the impact of land use activities: a case study from several catchments in Malaysia. Ph.D Thesis, University of Nottingham, Nottingham, UK.

Molls, T., & Chaudhry, M. H. (1995). Depth-averaged open-channel flow model. Journal of Hydraulic Engineering, 121(6), 453–465.

Naji, M. A., Ghodsian, M., Vaghefi, M., & Panahpur, N. (2010). Experimental and numerical simulation of flow in a 90° bend. Flow Measurement and Instrumentation, 21(3), 292–298.

Ottevanger, W., Blanckaert, K., & Uijttewaal, W. S. J. (2012). Modeling of the daily rainfall-runoff relationship with artificial neural network. Journal of Hydrology, 369, 285–294.

Rajurkar, M. P., Kothyari, U. C., & Chaube, U. C. (2004). Processes governing the flow redistribution in sharp river bends. Geomorphology, 63–64, 45–55.

Ramamurthy, A., Han, S., & Biron, P. (2013). Three-dimensional simulation parameters for 90° open channel bend flows. Journal of Computing in Civil Engineering, 27(3), 282–291.

Rajurkar, M. P., Kothyari, U. C., & Chaube, U. C. (2004). Modeling of the daily rainfall-runoff relationship with artificial neural network. Journal of Hydrology, 285, 96–113.

Rozovskii, I. L. (1961). Flow of water in bends of open channels. Academy of Sciences of the Ukrainian SSR, Israel Program for Science Translation, Jerusalem, 1–233.

Sahu, M., Jana, S., Agarwal, S., & Khatua, K. K. (2011). Point form velocity prediction in meandering open channel using artificial neural network. 2nd International Conference on Environmental Science and Technology, 6, IACSIT Press, Singapore, 209–212.

Shamshirband, S., Gocic, M., Petkovic, D., Javidnia, H., Hafizah, S., Hamid, A., . . . Qasem, S. N. (2015). Clustering project management for drought regions determination: a case study in Serbia. Agricultural and Forest Meteorology, 200(15), 57–65.

Steffler, P. M., Rajartnam, N., & Peterson, A. W. (1985). Water surface change of channel curvature. Journal of Hydraulic Engineering, 111(5), 866–870.

Stoesser, T., Ruether, N., & Olsen, N. R. B. (2010). Calculation of primary and secondary flow and boundary shear stresses in a meandering channel. Advances in Water Resources, 33, 158–170.

Wang, W. C., Chau, K. W., Cheng, C. T., & Qiu, L. (2009). A comparison of performance of several artificial intelligence methods for forecasting monthly discharge time series. Journal of Hydrology, 374(3–4), 294–306.

Wang, W. C., Xu, D. M., Chau, K. W., & Lei, G. J. (2014). Assessment of river water quality based on theory of variable fuzzy sets and fuzzy binary comparison method. Water Resources Management, 28(12), 4183–4200.

Wu, C. L., & Chau, K. W. (2010). Data-driven models for monthly streamflow time series prediction. Engineering Applications of Artificial Intelligence, 23(8), 1350–1367.

Wu, C. L., Chau, K. W., & Li, Y. S. (2008). River stage prediction based on a distributed support vector regression. Journal of Hydrology, 358(1–2), 96–111.

Yakhot, V., & Orszag, S. A. (1986). Renormalization group analysis of turbulence I: basic theory. Journal of Scientific Computing, 1, 3–51.

Ye, J., & McCorquodale, J. A. (1998). Simulation of curved open channel flows by 3D hydrodynamic model. Journal of Hydraulic Engineering, 124(7), 687–698.

Yuhang, Z., & Wexin, H. (2009). Application of artificial neural network to predict the friction factor of open channel flow. Communications in Nonlinear Science and Numerical Simulation, 14, 2373–2378.

Zaji, A. H., & Bonakdari, H. (2015). Application of artificial neural network and genetic programming models for estimating the longitudinal velocity field in open channel junctions. Flow Measurement and Instrumentation, 41, 81–89.

Zeng, J., Constantinescu, G., Blanckaert, K., & Weber, L. (2008). Flow and bathymetry in sharp open-channel bends: experiments and predictions. Water resources research, 42(1–2), 1–12.

Zhang, M. L., & Shen, Y. M. (2008). Three dimensional simulation of meandering river based on 3-D k – ε (RNG) turbulence model. Journal of Hydrodynamics, 20(4), 448–455.

Zhou, G., Wang, H., Shao, X., & Jia, D. D. (2006). 2-D numerical simulation of flow in a curved open channel. Advances in Water Resources and Hydraulic Engineering, 3, 871–876.