National Center for Computational
Hydroscience and Engineering

CCHE2D Verification and Validation Tests
Documentation

August, 2001

Yafei Jia and Sam S.Y. Wang

School of Engineering
The University of Mississippi
University, MS 38677
CCHE2D Verification and Validation Tests

Documentation

Version 2.1

NCCHE-TR-2001-2

Project Director and Principle Investigator
Sam S.Y. Wang, Ph.D.
F.A.P. Barnard Distinguished Professor and Director
Center for Computational Hydroscience of Engineering
The University of Mississippi
University, MS 38677

Senior Investigator
Yafei Jia, Ph.D.
Research Associate Professor
Center for Computational Hydroscience of Engineering
The University of Mississippi
University, MS 38677

August 2001
Abstract

This report is a collection of results from many tests conducted using the CCHE2D model. It is a complementary document of the model’s technical report (NCCHE TR2001-1: CCHE2D: a two-dimensional hydrodynamic and sediment transport model for unsteady open channel flows over loose bed). Verification tests are based on cases of analytical solution, physical model data and natural open channel flows.

This report intends to provide the prospective users with some qualitative and quantitative information about the CCHE2D’s capabilities and limitations, to enable them to decide whether it is applicable to the problems they are planning to investigate. More results have been included in this version of the documentation than the previous one, including comparison with analytical solution, experiment data, field data for flow and sediment transport.

Due to the difficulty in obtaining physical model data, especially the ones collected in the field, some feasibility studies presented without physical model data comparison. Users may use their knowledge and experience to assess the reasonableness of the simulated results, however. Additional verification results shall be published in the professional journals and conference proceedings as soon as they are completed in the future.
# Table of Contents

<table>
<thead>
<tr>
<th>Title</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>TABLE OF CONTENTS</td>
<td>1</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>4</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>8</td>
</tr>
<tr>
<td>CHAPTER 1</td>
<td>9</td>
</tr>
<tr>
<td>INTRODUCTION</td>
<td>9</td>
</tr>
<tr>
<td>CHAPTER 2</td>
<td>11</td>
</tr>
<tr>
<td>VERIFICATION BY ANALYTICAL SOLUTION</td>
<td>11</td>
</tr>
<tr>
<td>2.1. Verify the second order terms of the flow solver with parabolic profile because</td>
<td>11</td>
</tr>
<tr>
<td>2.2. Verify the transport equations for general scalar transport study</td>
<td>15</td>
</tr>
<tr>
<td>CHAPTER 3</td>
<td>20</td>
</tr>
<tr>
<td>VERIFICATION BY EXPERIMENTAL DATA (SUBCRITICAL FLOW)</td>
<td>20</td>
</tr>
<tr>
<td>3.1. Flow around a spur dike</td>
<td>20</td>
</tr>
<tr>
<td>3.1.1 Boundary conditions</td>
<td>21</td>
</tr>
<tr>
<td>3.2. Flow in a channel of sudden expansion</td>
<td>24</td>
</tr>
<tr>
<td>3.3. Flow in a 180° U-shaped channel</td>
<td>26</td>
</tr>
<tr>
<td>3.3.1 Boundary conditions</td>
<td>26</td>
</tr>
<tr>
<td>3.4. Flow in sine-generated curved channels</td>
<td>29</td>
</tr>
<tr>
<td>3.4.1 Channel with $\theta_o = 30^\circ$</td>
<td>30</td>
</tr>
</tbody>
</table>
3.4.1.1. Boundary conditions .................................................................................. 30
3.4.2. Channel with $\theta_o = 110^\circ$ ................................................................. 31
3.4.2. Boundary conditions .................................................................................. 31
3.5. Flow in a Compound Channel .................................................................... 33
3.5.1 Boundary conditions .................................................................................. 35

CHAPTER 4 ............................................................................................................ 37

VERIFICATION BY EXPERIMENTAL DATA (SUPERCRITICAL FLOW) ...... 37
4.1. Supercritical Flow in a Contraction Channel .............................................. 37
  4.1.1. Boundary conditions ............................................................................. 38
4.2. Subcritical to Supercritical Flow Transition .............................................. 40
  4.2.1. Flat bed with a contraction channel ..................................................... 41
    4.2.1.1. Boundary conditions ..................................................................... 41
  4.2.2. Uneven bed and contraction-expansion channel .................................. 43
    4.2.2.1. Boundary conditions ..................................................................... 44
4.3. Subcritical-Supercritical-Subcritical Transition ......................................... 45

CHAPTER 5 ............................................................................................................ 46

VERIFICATION OF SEDIMENT TRANSPORT MODEL ................................. 46
5.1. Aggradation in Alluvial Channels ............................................................... 46
  5.1.1. Boundary conditions ............................................................................ 46
5.2. Degradation in Alluvial Channels .............................................................. 48
  5.2.1. Boundary conditions ............................................................................ 48
5.3. Bed Morphological Change in Meandering Rivers ................................. 49
5.4. Simulation of Alluvial River Channel Migration ....................................... 52
5.5. Suspended Sediment Transport Simulation in River Nechar, Germany 53

CHAPTER 6 ............................................................................................................ 57

FEASIBILITY STUDIES .......................................................................................... 57
6.1. Flow around Spur Dikes in Natural Channel ............................................. 57
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2.</td>
<td>Unsteady Flow in a Natural River Channel</td>
<td>58</td>
</tr>
<tr>
<td>6.3.</td>
<td>Unsteady Flow Simulation in River Nechar, Germany</td>
<td>61</td>
</tr>
<tr>
<td>6.4</td>
<td>Simulation of Flow in East Fork River</td>
<td>65</td>
</tr>
<tr>
<td>6.5</td>
<td>Simulation of Flows Associated with In-Stream Hydraulic Structures</td>
<td>68</td>
</tr>
<tr>
<td>6.5.1</td>
<td>Red River Lock &amp; Dam structure</td>
<td>68</td>
</tr>
<tr>
<td>6.5.2</td>
<td>Mississippi River, Old River Complex</td>
<td>69</td>
</tr>
<tr>
<td>6.5.3</td>
<td>Simulation of flow with spur dikes and submerged weirs in Victoria Bendway, Mississippi</td>
<td>72</td>
</tr>
<tr>
<td>6.5.4</td>
<td>Simulation of flow in the channel downstream of Wanan Reservoir-A case with dike field</td>
<td>78</td>
</tr>
</tbody>
</table>

**CHAPTER 7** | CONCLUSION | 80

**REFERENCES** | 81
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 2.1</td>
<td>Comparison between analytical solution and simulations. ................. 13</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Development of laminar flow in a channel. ................................... 13</td>
</tr>
<tr>
<td>Figure 2.3</td>
<td>Comparison of analytical solution with simulations with different mesh</td>
</tr>
<tr>
<td></td>
<td>distributions ............................................................................... 15</td>
</tr>
<tr>
<td>Figure 2.4</td>
<td>Analytical solution of equation (2.6), c* ................................ 16</td>
</tr>
<tr>
<td>Figure 2.5</td>
<td>Simulated convection –diffusion of concentration ............................ 17</td>
</tr>
<tr>
<td>Figure 2.6</td>
<td>Comparisons of analytical solution and simulated concentration .......... 17</td>
</tr>
<tr>
<td></td>
<td>distribution in a long channel. Concentration is discharged near a bank.</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>(a) Mesh system used for the spur dike (b) Detail around the spur. ....... 21</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Comparisons of the simulations with the experimental data: (a) Mixing</td>
</tr>
<tr>
<td></td>
<td>length eddy viscosity, total slip boundary condition; (b) Mixing length</td>
</tr>
<tr>
<td></td>
<td>eddy viscosity, wall function boundary condition; and (c) Eddy viscosity</td>
</tr>
<tr>
<td></td>
<td>due to bed shear, wall function boundary condition. ........................ 23</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>Sketch of the flow pattern and the flume of sudden expansion ............ 24</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Comparisons of the simulated and measured flow field in the channel of</td>
</tr>
<tr>
<td></td>
<td>sudden expansion, a: Q=0.01815m³/s, b: Q=0.03854m³/s ..................... 25</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>Mesh system of the 180° U-shaped channel. .................................... 27</td>
</tr>
<tr>
<td>Figure 3.6</td>
<td>Comparison of simulation and measurement. .................................... 28</td>
</tr>
<tr>
<td>Figure 3.7</td>
<td>Comparison of the computed and measured superelevation .................... 29</td>
</tr>
<tr>
<td>Figure 3.8</td>
<td>Computational mesh for case with $\theta_o=30^\circ$ ....................... 30</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>3.9</td>
<td>Comparison of the computed and measured velocity for sine-generated channel: $\theta_o = 30^\circ$</td>
</tr>
<tr>
<td>3.10</td>
<td>Computational mesh for case with $\theta_o = 110^\circ$</td>
</tr>
<tr>
<td>3.11</td>
<td>Comparison of the computed and measured velocity for sine-generated channel: $\theta_o = 110^\circ$</td>
</tr>
<tr>
<td>3.12</td>
<td>Definition sketch of a compound channel cross-section</td>
</tr>
<tr>
<td>3.13</td>
<td>Mesh and water depth distributions a compound channel cross-section</td>
</tr>
<tr>
<td>3.14</td>
<td>Comparison of measured and simulated longitudinal velocity in a compound channel</td>
</tr>
<tr>
<td>4.1</td>
<td>Mesh system for standing wave case</td>
</tr>
<tr>
<td>4.2</td>
<td>Measured and computed water surface elevation pattern; (a) data (b, c) simulation</td>
</tr>
<tr>
<td>4.3</td>
<td>Comparison of experimental data with numerical simulations; (a) along central line (b) along side wall</td>
</tr>
<tr>
<td>4.4</td>
<td>Mesh system for subcritical to supercritical flow case</td>
</tr>
<tr>
<td>4.5</td>
<td>Comparison of experimental data (a) with numerical simulation (b)</td>
</tr>
<tr>
<td>4.6</td>
<td>Comparison of simulated and measured surface elevation along center line</td>
</tr>
<tr>
<td>4.7</td>
<td>Mesh system for the Parshall flume</td>
</tr>
<tr>
<td>4.8</td>
<td>Comparisons of simulation with data for a Parshall flume; (a) Velocity (b) Water surface</td>
</tr>
<tr>
<td>4.9</td>
<td>Subcritical-supercritical-subcritical regime transition</td>
</tr>
<tr>
<td>5.1</td>
<td>Verification of sediment transport model using aggradation experiments</td>
</tr>
</tbody>
</table>
Figure 5.2  Verification of sediment transport model using degradation experiments.  49
Figure 5.3  Simulated bed change in a meandering channel (early stages) .......... 50
Figure 5.5  Simulated channel migration. The channel is of laboratory scale. ........ 52
Figure 6.1  Simulated flow field around spur dikes in Hotophia Creek ............... 57
Figure 6.2  Bed topography of the simulated reach of Hotophia Creek ............... 58
Figure 6.3a Simulated unsteady flow in Hotophia Creek (rising stage). ............... 59
Figure 6.3b Simulated unsteady flow in Hotophia Creek (falling stage). .............. 60
Figure 6.4  Simulated flow pattern in the Neckar River. Moving boundary technique makes the unsteady flow simulation possible. Detailed flow circulation on the sallow areas driven by the main-channel flow can be observed. ........................................ 62
Figure 6.5  Comparison of the simulated and measure water surface elevation for three floods in River Neckar, Germany ................................................................. 63
Figure 6.6  Comparison of simulated and measured surface elevation in a flood event along the channel of River Nechar, Germany ....................................................... 64
Figure 6.7  Simulated flow pattern at downstream .............................................. 69
of Lock&Dam 2, Red River .................................................................................. 69
Figure 6.8  Simulation of channel flow in Mississippi, Old River Complex .......... 71
Figure 6.9  Simulated flow at the junction of the intake channel to the hydropower plant and Mississippi ................................................................. 72
Figure 6.10 Victoria Bendway and the survey lines, depth-averaged velocities are shown 74
Figure 6.11 Comparison of the computed depth-averaged velocity (m/s) and that of measured (June, 11, 1998). Black points are velocities, red points are reported errors in the measured data ................................................................. 75
Figure 6.12  Velocity magnitude (m/s) and vectors in the Victoria Bendway without submerged weirs, CCHE2D........................................................................................................... 77

Figure 6.13. Simulated flow field in Wanan River with multiple dikes protecting banks 78

Figure 6.14 Simulated flow in Wanan River close to the Dam of Wanan Reservoir ...... 79
# List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2.1: Parameters for the laminar flow case</td>
<td>12</td>
</tr>
<tr>
<td>Table 3.1: Flow conditions of the physical experiment</td>
<td>20</td>
</tr>
<tr>
<td>Table 3.2: Flow conditions for the sudden expansion channel</td>
<td>24</td>
</tr>
<tr>
<td>Table 3.3: Flow conditions for the 180° U-shaped channel</td>
<td>26</td>
</tr>
<tr>
<td>Table 3.4: Flow conditions for sine-generated channel $\theta_o=30^\circ$</td>
<td>30</td>
</tr>
<tr>
<td>Table 3.5: Flow conditions for sine-generated channel $\theta_o=110^\circ$</td>
<td>31</td>
</tr>
<tr>
<td>Table 3.6: Flow conditions of compound channel experiment</td>
<td>34</td>
</tr>
<tr>
<td>Table 4.1: Flow conditions for standing wave case</td>
<td>37</td>
</tr>
<tr>
<td>Table 4.2: Flow conditions for subcritical to supercritical flow case</td>
<td>41</td>
</tr>
<tr>
<td>Table 4.3: Flow conditions for subcritical to supercritical flow case</td>
<td>43</td>
</tr>
<tr>
<td>Table 5.1: Sediment and initial flow conditions (Sony, 1981)</td>
<td>46</td>
</tr>
<tr>
<td>Table 5.2: Sediment and initial flow conditions (Newton, 1951)</td>
<td>48</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Verification and validation are the most important steps in the numerical model developing processes. After the model is implemented, it is the only way to tell if the numerical model is free of bugs and capable of predicting the physical phenomena it has been designed for. Inadequate predictions are signs of problems in the model. On the other hand, numerical models are product of our limited understanding to the real world physics, assumptions and approximations to reach simplicity. For example, CCHE2D is a depth-integrated two dimensional model; it should not be used in situations where the three dimensional flow effects on the solution are significant. In the application of a numerical model to practical problems, therefore, approximate solution and trend should be expected instead of perfect agreement. In addition, one has to evaluate the uncertainties for the measured data before any conclusions on the results comparison.

This report has collected many verification and validation tests conducted before the release of the last version of this model. Many new verification and validation tests have been conducted since then, and we have tried to include as many new test cases as possible to this report, especially applications of unsteady sediment transport, bank erosion and flow simulations in natural rivers and waterways with hydraulic structures. One can tell through these test cases that the prediction of the CCHE2D model agrees quite well with a lot of measurements from physical experiments including supercritical flow and channel morphological changes. Most of the verifications are made by comparison of the simulation with physical experimental data because analytical solution and field measurement data are rate. These verifications can greatly increase the user’s confidence of applying this model to their practical problems.

Details of the flow conditions and boundary conditions of these simulations are described; it is possible to repeat these simulations with these conditions.
The model’s development and verification/validation are at the National Center for Computational Hydroscience and Engineering under the supervision of Dr. Sam S.Y. Wang. The verification/validation cases are prepared by Dr. Yafei Jia and other members of NCCHE. Many NCCHE scholars and students have participated in the verification/validation process including Drs. Y. Zhang, G.H. Duan, Y. Xu, Ms. Y. Kai, and Mr. G.C. Song. Their efforts are deeply appreciated.
Chapter 2

Verification by analytical solution

2.1. Verify the second order terms of the flow solver with parabolic profile

Comparison of the numerical solution with an analytical solution has been conducted in this chapter. Because it is very difficult to obtain a general analytical solution for the non-linear momentum equations and the continuity equation, only one case of comparison was made.

The analytical solution of a laminar flow in a straight channel with constant slope is obtained by solving the equation

\[
\frac{\partial^2 u}{\partial y^2} = k \frac{\partial p}{\partial x}
\]  

(2.1)

where \( y \) is the transversal coordinate, and \( u \) is the longitudinal velocity component. This equation is obtained by simplifying the original depth-integrated momentum equations under above boundary conditions. The analytical solution reads

\[
u = 4u_m y (1 - y)
\]  

(2.2)

where \( u_m \) is the maximum velocity along central line, and the total discharge is
\[ Q = \frac{2}{3} u_w h \] \hspace{1cm} (2.3)

\( h \) is water depth. This is because the depth-averaged model needs water depth, any arbitrary \( h \) should gave the same results. Table 2.1 shows parameters used in the verification test.

**Table 2.1: Parameters for the laminar flow case**

<table>
<thead>
<tr>
<th>B (m)</th>
<th>h (m)</th>
<th>( v \times 10^4 ) (m(^2)/s)</th>
<th>( S_o )</th>
<th>( U_m ) (m/s)</th>
<th>Q (m(^3)/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>0.8</td>
<td>1.225</td>
<td>1/10000</td>
<td>0.5</td>
<td>0.2667</td>
</tr>
</tbody>
</table>

For this particular verification case, only the second order terms in the momentum equations and continuity equation are activated. Bed shear stress terms are turned off, and non-slip boundary conditions are used along the wall boundaries. The initial water surface elevation was an arbitrary constant value and bed roughness (Manning's coefficient) is set to zero. The pressure \( p \) in the equation (2.1) is represented by solved water surface elevation. The following figure shows the comparison of calculated results and the analytical solution. Discharge distribution corresponding to the analytical solution is prescribed at the inlet section. It is seen that all the numerical solutions with different number of nodes in cross-section (from 5 to 21) agree with the analytical solution Figure 2.1. Because the interpolation function of this finite element model is quadrilateral, it is not surprising that different meshes produced the same solution. This is an indication that the numerical model is free of errors in mathematical derivation, numerical calculation and program implementation.
The next test of this model is to simulate the development of the laminar boundary layer. The boundary condition prescribed at the inlet section is constant specific discharge, resulting in a constant velocity distribution, and the computational results are shown in Figure 2.2. The solid curve in the figure is the analytical solution (2.2), the velocity
distribution changes rapidly near the inlet and then gradually approaches the desired curve.

In the figures shown, the mesh size in the transversal direction was constant. Sensitivity tests were conducted by changing the mesh spacing in the transversal direction systematically (shifting nodes towards the side walls). It was found that for irregular mesh (the mesh size increased towards to the center line of the channel with a constant ratio) the numerical solutions have almost no change. Figure 2.3 shows the results of using different mesh ratios from 1.1 to 3.0, no significant difference is observed either between the analytical solution and simulation results or between the simulations results themselves.

These tests verified that the numerical model converges to the analytical solution when the model solves the same problem as the differential equation of the analytical solution. It also tells us that the method is not sensitive to irregular mesh when the spacing ratio is less than 3.0. It should be aware that this result was obtained using a linear problem and can only be used as a reference. For applications of non-linear problems, the maximum spacing ratio 2.0 is recommended.
2.2. Verify the transport equations for general scalar transport study

It is well known that any diluted substance moving water flows is governed by the transport equation

$$\frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} + v \frac{\partial c}{\partial y} - \beta \nu_t \left[ \frac{\partial}{\partial x} \left( \frac{\partial c}{\partial x} \right) - \frac{\partial}{\partial y} \left( \frac{\partial c}{\partial y} \right) \right] = S$$

(2.4)

where $c$ is the concentration of the substance, $u$ and $v$ are velocity components in x and y directions, $\nu_t$ is the turbulent eddy viscosity and $\beta$ is the Schmidt number to convert $\nu_t$ to diffusivity for the transport of this substance. $S$ is a source term depending on the generation, consumption of the material in the flow.
Analytical solution for equation (2.4) is available under simplified conditions: 

\( u = \text{constant}, \ v = 0, \ b \nu = \text{constant}, \ S = 0, \) channel width \( B = \infty, \) and \( \partial^2 / \partial x^2 = 0 \)

The last condition is to neglect longitudinal diffusion. The solution after such simplification would be (Graf, 1998)

\[
c_u(x, y) = \frac{G_0}{h \sqrt{4 \pi b \nu t}} e^{-\frac{y^2}{4 b \nu t}}
\] (2.5)

where \( G_0 = \frac{M_0}{t} \) is the mass flow of a linear source (kg/s) distributed over the flow depth \( h. \) For a solution with finite channel width, boundary condition \( \partial c / \partial y = 0 \) along side walls is applied and the solution has the form

\[
c(x, y) = c_u(x, y + y_0) + c_u(x, y - y_0) + \sum_{n=1}^{N} c_u(x, 2nB \pm y \pm y_0)
\] (2.6)

The subscript \( u \) indicate unbounded solution \( (B = \infty) \)

Figure 2.4 Analytical solution of equation (2.6), \( c^*. \)
Figure 6.4 shows the analytical solution of the equation (2.6). Where $y^* = y/B$, $x^* = x \cdot \frac{\beta v_t}{u B^2}$, and $c^* = C/C_m$. $C$ is the depth averaged concentration; $C_m$ is the final concentration after the transversal diffusion has completely mixed the concentration in the entire cross-section; and $Q$ is the flow discharge. Figure 6.5 shows the simulated concentration distribution using CCHE2D transport model in the same channel with the same flow and boundary conditions.

Equilibrium or complete mixing occurs at $x^* = 1$ where $c$ would no longer be the function of $x$. The simulated concentration distribution in literal direction is almost the same as the analytical solution, the propagation to downstream direction is slightly less than the analytical solution. One should note the absolute distance from the concentration discharge inlet to the fully mixed downstream is more than 100km, the mesh density towards to the downstream decreases dramatically and it was very sparse in the downstream reach. This could contribute to the above discrepancy. The overall agreement is quite satisfactory.

Figure 2.5  Simulated convection –diffusion of concentration
Figure 6.6 show the simulation of a different case: the upstream concentration is releases near one bank of the channel. The diffusion brings the concentration across the entire cross section to the other bank. The error for this simulation is a little higher than that if the symmetric concentration case, due to that the high concentration is close to the wall boundary. The boundary condition at the wall $y^*=0$ is $\frac{\partial c}{\partial y} = 0$. It was found that the third order finite difference operator is necessary to provide accurate boundary condition.
Figure 2.6  Comparisons of analytical solution and simulated concentration distribution in a long channel. Concentration is discharged near a bank.
Chapter 3

Validations by experimental data
(subcritical flow)

3.1. Flow around a spur dike

These verification tests compare the solution of the model with experimental data measured in a straight flume with a thin plate spur dike (Rajaratnem and Nwachukwa 1983). A thin plate of 3mm thick projecting perpendicular made the spur dike to the vertical side wall.

The channel bed and the vertical walls (banks) are 'smooth'. The bulk flow data of the flume test are listed in the Table 3.1.

Table 3.1: Flow conditions of the physical experiment

<table>
<thead>
<tr>
<th>Q  $(m^3/s)$</th>
<th>V  $(m/s)$</th>
<th>H  $(m)$</th>
<th>B  $(m)$</th>
<th>$F_r$</th>
<th>D  $(m)$</th>
<th>R</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0453</td>
<td>0.26</td>
<td>0.189</td>
<td>0.915</td>
<td>0.19</td>
<td>0.152</td>
<td>12</td>
</tr>
</tbody>
</table>

where: $Q = \text{discharge}$, $V = \text{mean velocity}$, $H = \text{mean water depth}$, $B = \text{channel width}$, $F_r = \text{Froude number}$, $D = \text{length of dike}$, and $R = \text{re-attachment } /D$. 
3.1.1 Boundary conditions

- **Inlet section:** Constant total discharge \( Q \) is prescribed.
- **Outlet section:** Constant water surface elevation \( h = 0.189m \) was specified.
- **Roughness on the bed:** This was a 'smooth' channel, roughness on the bed can be assigned a small value. In fact, Manning's coefficient \( n = 0.01 \) or roughness height \( k_r \leq 0.001m \) shall produce the same results.
- **Wall boundary:** Both total slip and wall function were applied to this case separately.

Figure 3.1a shows the mesh system used for this simulation and Figure 3.1b shows the details around the spur dike. In order to have good solution, the mesh near the tip of the dike was very fine.

![Mesh system used for the spur dike](image)

![Detail around the spur](image)

The velocities at two levels \( (z/H=0.03 \) and \( z/H=0.85 \)) and in 4 sections \( (x/D=2, 4, 6, 8) \) were measured across the recirculation zone behind the dike. The data indicated that the flow in the recirculation zone had almost constant negative velocity at these four sections; and there was a maximum velocity peak value appears just outside the shear layer especially near the bed; and the velocity profiles were quite uniform outside the shear layer in the main flow region. Since the CCHE2D model simulates the depth
averaged flow, the data at the level $z/H=0.03$ was considered too close to the bed, and the data measured at level $z/H=0.85$ was used for comparison.

The flow recirculation behind the dike is closely related to the shear layer. The eddy viscosity model used to simulate the problem plays an important role in the accuracy of the prediction because the Reynolds stress in the model is proportional to the eddy viscosity. Both eddy viscosity models (CCHE-TR-97-1) in the CCHE2D model were used for this simulation.

Figure 3.2 shows the comparison. The depth averaged mixing length model was applied (Fig. 3.2a and 3.2b) and the solutions agreed very well with the measurement. Fig. 3.2c is the result of using depth averaged parabolic eddy viscosity. It can be seen that the mixing length model results were much better than that of the parabolic model, although the latter had good prediction in the main channel, the velocity in the recirculation zone was much lower than the measurement. The difference between the results of wall function and total slip boundary condition was not large except near the wall.
Figure 3.2 Comparisons of the simulations with the experimental data: (a) Mixing length eddy viscosity, total slip boundary condition; (b) Mixing length eddy viscosity, wall function boundary condition; and (c) Eddy viscosity due to bed shear, wall function boundary condition.
3.2. Flow in a channel of sudden expansion

In addition to the dike case, the CCHE2D model was validated by using a case of a channel with a sudden expansion in width. The velocity data were measured in a flume (Xie, 1994) for two flow discharges along 11 cross-sections from the expansion section to downstream of the recirculation zone. The flume was of concrete bed and walls with fixed bed slope. All solid surfaces have been considered as hydraulic smooth in the computation. The geometry of the channel and the flow conditions are shown in Figure 3.3 and Table 3, respectively. The measured velocity distributions indicate that the recirculation length in these two experiments is approximately the same.

![Figure 3.3 Sketch of the flow pattern and the flume of sudden expansion](image)

**Table 3.2:** Flow conditions for the sudden expansion channel

<table>
<thead>
<tr>
<th>Discharge (m$^3$/s)</th>
<th>Width (m)</th>
<th>Depth (m)</th>
<th>Step Height (m)</th>
<th>Slope</th>
<th>Approach main velocity (m/s)</th>
<th>Approach Froude Number</th>
<th>Recirculation Length (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01815</td>
<td>1.2</td>
<td>0.101</td>
<td>0.6</td>
<td>1/1000</td>
<td>0.30</td>
<td>0.30</td>
<td>4.60</td>
</tr>
<tr>
<td>0.03854</td>
<td>1.2</td>
<td>0.105</td>
<td>0.6</td>
<td>1/1000</td>
<td>0.60</td>
<td>0.60</td>
<td>4.60</td>
</tr>
</tbody>
</table>
The $k-\varepsilon$ turbulence closure model was applied to simulate this case. Figure 3.4 shows the comparisons of the simulated and measured velocity in 11 and 9 cross-sections downstream of the expansion section for these two cases. The two simulations used the same numerical mesh, which is of 35x61 in the expansion part and 15x19 in the approaching channel part. As one can tell, the overall agreement between the data and the simulation results are excellent. The flow in the main channel as well as in the recirculation zone follows closely to the trend of the measured data. The computed velocity distribution in the shear layer has the same slope as those measured. The reattachment length indicated by the data is about 4.6m (Figure 3.4), equivalent to 7.83 $\Delta H$, slightly longer than those measured in the confined condition (Chen, 1985). The predicted reattachment point is approximately at the same location. The inverse velocity in the middle of the recirculation zone is approximately a constant, similar to those measured in the spur dike case (Rajaratnam, and Nwachukwu, 1983). The simulated strength of the inverse flow is comparable to the measurement however, for the case of smaller discharge, it is weaker than the measured ones near the recirculation point.

Figure 3.4  Comparisons of the simulated and measured flow field in the channel of sudden expansion, a: $Q=0.01815m^3/s$, b: $Q=0.03854m^3/s$
3.3. Flow in a $180^\circ$ U-shaped channel

The most commonly seen river channel pattern in nature is meandering. In order to test the model's capability to simulate the flow in meandering channels, the verification was conducted first for a U-shaped $180^\circ$ channel. This case is simple in the sense that the channel has a constant curvature. Detailed horizontal velocity components had been measured in this flume in many cross-sections, and each section has many verticals with many measurement points along each vertical (De Vriend, 1979). In fact this data set is very good for three dimensional model verification, the measured velocity field was averaged along water depth for the purpose of verification for two dimensional models.

Table 3.2 shows the flow conditions of the experiment. The experimental channel was quite large, its width was 1.7 meter, the radius of the central line curvature was 4.25 meters. The same conditions are applied to the CCHE2D model and simulation was conducted.

**Table 3.3:** Flow conditions for the $180^\circ$ U-shaped channel

<table>
<thead>
<tr>
<th>Q</th>
<th>$D_{50}$</th>
<th>B</th>
<th>$h_m$</th>
<th>$s_b$</th>
<th>$u_m$</th>
<th>$Re_*$</th>
<th>$F_r$</th>
<th>R</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m^3/s$</td>
<td>(mm)</td>
<td>(m)</td>
<td>(m)</td>
<td></td>
<td>(m/s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.180</td>
<td>1.0</td>
<td>1.7</td>
<td>0.1953</td>
<td>0.542</td>
<td>1513</td>
<td>0.392</td>
<td>5.1</td>
<td></td>
</tr>
</tbody>
</table>

3.3.1 Boundary conditions

Inlet section: The velocity measurement at the first upstream section, which is close to the simulation inlet section, was used to prescribe the specific discharge distribution at the inlet section. In fact, uniform discharge distribution should be
equally good in this situation because the measured distribution was quite uniform (Figure 3.4).

- Outlet section: Constant water surface elevation was used.
- Bed roughness: $k_r \leq 0.001m$.
- Wall boundary: Wall function was applied along both vertical walls of the channel.

Figure 3.3 is the computational mesh used for this verification test. Higher mesh density was placed in the curved part of the channel and along the walls. Total number of nodes was 23x49.

Figure 3.5  Mesh system of the 180° U-shaped channel

Figure 3.4 shows the comparison between the measured velocity and the simulation. As mentioned above, the measured velocities had been averaged along the depth. All the
sections with measured data are present in the figure. It obvious through this figure that the CCHE2D model captured all the features of the depth averaged flow in this channel.

![Diagram of water surface elevation in curved channel with experimental data and CCHE2D predictions.](image)

**Figure 3.6** Comparison of simulation and measurement.

Water surface elevation in curved channels has characteristics of superelevation. The water surface elevation at the outer bank side is higher than that along the inner bank due to the centrifugal force. Figure 3.5 shows the comparison of computed and the measured free surface elevations. Although discrepancy exists, the predicted superelevation is less than the measured, the trend predicted agrees well with each other. One of the reasons that could lead to this discrepancy is that the secondary current in the cross section could not be simulated by the depth averaged model. It is well known that the secondary current plays an important role in building up the superelevation. It can be shown that three dimensional model could produce more realistic results (Jia and Wang, 1992).
3.4. Flow in sine-generated curved channels

Since natural river channel form is complicated, constant curvature channel only exist in laboratories, more realistic channel form ought to be used to verify numerical models. Among others, the sine-generated function has been widely used to represent the natural channel form:

$$\theta = \theta_0 \cos \left( \frac{2\pi l}{L} \right)$$

where $\theta$ is the angle between the channel curve and the downstream direction. $\theta_0$ is the maximum angle at the cross over section, $l$ is the longitudinal distance and $L$ is the wave length of the meander. This formula best describes the continuous variation of curvature change of meandering rivers. Verification conducted using this type of channel is based on two sets laboratory data measured by Dr. de Silva (1995).
3.4.1. Channel with $\theta_o = 30^\circ$

The flow conditions for the first set of data are shown in the Table 3.3. This is a relatively small channel, the water depth is set very small in order to satisfy shallow water condition.

**Table 3.4:** Flow conditions for sine-generated channel $\theta_o = 30^\circ$

<table>
<thead>
<tr>
<th>Q (l/s)</th>
<th>$D_{50}$ (mm)</th>
<th>B (m)</th>
<th>$h_m$ (m)</th>
<th>$s_b$</th>
<th>$u_m$ (m/s)</th>
<th>$Re_*$</th>
<th>$F_r$</th>
<th>$B/h_m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.10</td>
<td>2.2</td>
<td>0.4</td>
<td>3.2</td>
<td>1/1000</td>
<td>6.4</td>
<td>5250</td>
<td>0.086</td>
<td>12.5</td>
</tr>
</tbody>
</table>

3.4.1.1. Boundary conditions

- Inlet section: Uniform discharge distribution.
- Outlet section: Constant water surface elevation.
- Bed roughness: Sediment size $d_{50}$ was used as roughness hight.
- Wall boundary: Wall function is applied along the vertical walls.

Figure 3.6 shows the computational mesh for this case. For simplicity, the mesh sizes were uniform in both longitudinal and transversal direction (17x42).

Figure 3.7 compares the simulation results and the measurements. In the near wall region of convex bank velocity is slightly under estimated, almost perfect agreement is observed.
elsewhere. The flow has to accelerate when it is approaching the convex bank, which forces the flow to change direction. More resolution in this region may be necessary to capture the near wall acceleration more accurately.

![Diagram of flow and data comparison](image)

Figure 3.9 Comparison of the computed and measured velocity for sine-generated channel: $\theta_o = 30^\circ$.

### 3.4.2. Channel with $\theta_o = 110^\circ$

As channel meandering proceeds, meander bends become more and more curved, highly curved meander river can be seen often in the nature. The curvature change in this case is much more rapid than the previous case. Verification with this set of data has more practical importance. The flow conditions for this experiment are shown in Table 3.4

<table>
<thead>
<tr>
<th>Q</th>
<th>$D_{50}$</th>
<th>B</th>
<th>$h_m$</th>
<th>$s_b$</th>
<th>$u_m$</th>
<th>$Re_\ast$</th>
<th>$F_r$</th>
<th>$B / h_m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.10</td>
<td>2.2</td>
<td>0.4</td>
<td>3.0</td>
<td>1/1120</td>
<td>16.7</td>
<td>5000</td>
<td>0.095</td>
<td>13.3</td>
</tr>
</tbody>
</table>

### Table 3.5: Flow conditions for sine-generated channel $\theta_o = 110^\circ$

### 3.4.2. Boundary conditions

- Inlet section: Uniform discharge distribution.
- Outlet section: Constant water surface elevation.
- Bed roughness: Sediment size $d_{s0}$ was used as roughness height.
- Wall boundary: Slipness coefficient = 0.90.

Figure 3.8 shows the computational mesh for this case. This mesh is again uniform in both longitudinal and transversal directions (17x106). Calculated results and the measured experimental data are presented in Figure 8.9. The agreement is satisfactory in general, however, not good as that for the case with $\theta_0=30^\circ$. Differences are relatively high at inner bank when the flow is approaching the bend and at the outer bank when the flow is leaving the bend. The best agreement occurs in the band and the largest differences appear in the cross-over zone. Strong sinuosity appears to be the cause of the difference pattern, further studies are needed.

![Computational mesh for case with $\theta_0=110^\circ$.](image)

Figure 3.10 Computational mesh for case with $\theta_0=110^\circ$. 
3.5. Flow in a compound channel

Verification of numerical models using compound channel case is also important, because most natural rivers have compound cross-sectional form: a main channel with one or two flood planes. Figure 3.10 defines a simplified compound channel cross-section with one flood plane. Because the flow depth in the channel has distinctive difference (D for main channel, d for flood plane), the flow velocities in these two parts also differ greatly. In addition, the intersection between these two parts of the channel are relatively narrow, strong shear flow is usually developed near the intersection. Verification test like this shall tell us if the model can handle the shear flow at the intersection and the flow distribution in the two parts of the channel.
This verification test simulated the compound channel flow, the physical experimental data was measured by Rajaratnam and Ahmadi (1981). The flume was about 1.2m wide and 18.3 m long. 12.2m of the channel was simulated to save computing time. Figure 3.11 demonstrates the cross section form of the particular experiment with enlarged vertical size. For the convenience of the computation, original sharp step of the flood plane at 70 cm has been changed to a steep slope. The total water depth at the main channel side was about $D \approx 11.5\text{cm}$, while in the flood plane side was $d \approx 11.5\text{cm}$. This is the only case with complete data from which depth averaged velocity could be calculated. The vertical lines in this figure represent the mesh distribution in the transversal direction. Fine grids are distributed near the intersection and side walls. The longitudinal mesh size was a constant (0.2m). Table 3.5 shows flow conditions of this experiment.

**Table 3.6: Flow conditions of compound channel experiment**

<table>
<thead>
<tr>
<th>Expt. (#)</th>
<th>D (cm)</th>
<th>d (cm)</th>
<th>h (cm)</th>
<th>B (cm)</th>
<th>b (cm)</th>
<th>Q (m$^3$/s)</th>
<th>$S_b \times 10^3$</th>
<th>$F_r$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.28</td>
<td>1.52</td>
<td>9.75</td>
<td>71.1</td>
<td>50.8</td>
<td>0.027</td>
<td>0.450</td>
<td>0.37</td>
</tr>
</tbody>
</table>
3.5.1 Boundary conditions

- Inlet section: Initially, discharge distribution prescribed at the inlet section was a wild guess. The distribution near the outlet was than used at the inlet after the first time simulation has reach steady state approximately.
- Outlet section: Constant water surface elevation.
- Bed roughness: Channel bed was smooth.
- Wall boundary: Both wall function and slipness adjustment options were tested, results shall be discussed later.

Figure 3.13   Mesh and water depth distributions a compound channel cross-section

Figure 3.12 shows the results of the simulation and experimental data. Depth averaged mixing length eddy viscosity model was used. The fine dot line was the model result using wall function, no other parameters were adjusted. General agreement of this line with the data seems good. The simulated velocity change near the intersection was steeper than the data, which means the depth averaged model has certain limitation in this sharp changing zone. Attempt trying adjusting eddy viscosity coefficient had been made to see if higher shear stress could improve this problem. However, this it was found the higher viscosity not only changes the distribution near the interface, but also elsewhere, the worsen the overall agreement (dash line). The next attempt was to adjust the boundary slipness to improve the agreement. As a matter of fact, the measurements shown very little resistance near the vertical wall of the main channel. Assigning the slipness coefficient as 0.85 could indeed improve the agreement.
Figure 3.14 Comparison of measured and simulated longitudinal velocity in a compound channel.
Chapter 4

Verification by experimental data
(supercritical flow)

Verification cases discussed in this chapter are related to supercritical flows. Capabilities of calculating supercritical flow is significant from practitioner's point of view, because supercritical flow could happen both in natural channel and near hydraulic engineering structures.

4.1. Supercritical flow in a contraction channel

The first verification test case is the experiment conducted by Coles and Shintaku (1943). Supercritical flow ($F_r = 4.0$) was released from the inlet boundary, and standing waves were generated due to the contraction. Table 4.1 shows the flow conditions:

<table>
<thead>
<tr>
<th>$Q$ $(m^3/s)$</th>
<th>$S_b$</th>
<th>$F_{r,inlet}$</th>
<th>$F_{r,outlet}$</th>
<th>$H_{inlet}$ $(m)$</th>
<th>$H_{outlet}$ $(m)$</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0451</td>
<td>0.0</td>
<td>4.0</td>
<td>2.05</td>
<td>0.0314</td>
<td>0.08</td>
<td>0.01</td>
</tr>
</tbody>
</table>

Figure 4.1 shows the mesh system used for the simulation. The geometry of the flume is also clearly defined in this figure.
4.1.1. **Boundary conditions**

- Inlet section: Total discharge and water surface elevation was specified.
- Outlet section: Open boundary condition was applied.
- Roughness on the bed: Bed surface was smooth, very small Manning's coefficient was used.
- Wall boundary: Total slip.

Figure 4.2 shows the measured water surface elevation pattern (a) and simulation results (b). Although differences exist, overall pattern of the predicted surface is similar to the measurement.
Comparisons of the water surfaces along the center line and one side wall of the flume are shown in Figure 4.3. Beside the results from CCHE2D model, those from other two numerical models are also shown (Molls et al 1995, Bhallamudi and Chaudhry, 1992). It can be seen, the trend predicted by the numerical models is close to the data lines. The predicted peak position in the center line appears downstream of the measured one, although the peak heights ear the same. Along the side wall, all the numerical model predicted a peak at the location of 1.7m, which is not shown in the data.
4.2. Subcritical to supercritical flow transition
4.2.1. Flat bed with a contraction channel

Flow regime transition from subcritical flow to supercritical flow had also modeled experimentally by Coles and Shintaku (1943). Flow conditions are shown in Table 4.2. Subcritical flow was released from the upstream inlet section and the flow regime changed to supercritical due to the contraction.

Table 4.2: Flow conditions for subcritical to supercritical flow case

<table>
<thead>
<tr>
<th>$Q$ ($m^3/s$)</th>
<th>$S_b$</th>
<th>$F_{r\text{inlet}}$</th>
<th>$F_{r\text{outlet}}$</th>
<th>$H_{\text{inlet}}$ (m)</th>
<th>$H_{\text{outlet}}$ (m)</th>
<th>n</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0451</td>
<td>0.0</td>
<td>0.315</td>
<td>1.55</td>
<td>0.175</td>
<td>0.09</td>
<td>0.01</td>
</tr>
</tbody>
</table>

Figure 4.4 shows the mesh system used for the simulation, which is similar to that used for the supercritical simulation. The geometry of the flume is also clearly defined in this figure.

![Image of mesh system](image)

Figure 4.4 Mesh system for subcritical to supercritical flow case.

4.2.1.1. Boundary conditions

- Inlet section: Total discharge was specified.
• Outlet section: Water surface elevation was applied.
• Roughness on the bed: Bed surface was smooth, very small Manning's coefficient was used.
• Wall boundary: Total slip.

Figure 4.5 shows the measured water surface elevation pattern (a) and simulation results (b). Again, overall pattern of the predicted surface is similar to the measurement. Important difference is that the curvatures of the measured contour lines are larger and more irregular than those of the computation results.

(a)

(b)

Figure 4.5 Comparison of experimental data (a) with numerical simulation (b).
Figure 4.6 shows the comparison of water surface elevation along center line, and the result of Molls et al (1995) is also included. The overall agreement is very encouraging.
4.2.2. Uneven bed and contraction-expention channel

Jian and McCorquodale (1997) simulated the supercritical outflow in a Parshall flume measured by Parshall (1926). This case was also simulated with the CCHE2D model. Flow conditions are shown in Table 4.3. Subcritical flow was released from the upstream inlet section and the flow regime changed to supercritical due to the contraction and the slope change.

Table 4.3: Flow conditions for subcritical to supercritical flow case

<table>
<thead>
<tr>
<th>Q (m³/s)</th>
<th>h_m (m)</th>
<th>u_m (m/s)</th>
<th>F_r</th>
<th>Nature of bed</th>
</tr>
</thead>
<tbody>
<tr>
<td>up</td>
<td>down</td>
<td>up</td>
<td>down</td>
<td></td>
</tr>
<tr>
<td>0.0451</td>
<td>0.12</td>
<td>0.103</td>
<td>0.4</td>
<td>1.375</td>
</tr>
</tbody>
</table>
Figure 4.7 shows the mesh system used for the simulation. The channel has a slope in the reach between contraction and expansion reaches and the expansion reach has an inverse bed slope. Other parts of the channel have no bed elevation changes.

4.2.2.1. Boundary conditions

- Inlet section: Total discharge was specified.
- Outlet section: Water surface elevation was applied.
- Roughness on the bed: bed surface was smooth, very small Manning's coefficient was used.
- Wall boundary: Total slip.

Comparisons of calculated velocity and water surface elevation are shown in Figure 4.8. Flow regime transfer occurs in the transition reach, and the flow keeps its supercritical
characteristics further downstream. It is seen, the agreements between the data and the solutions are good in general except some discrepancy in the transition reach of the channel.

4.3. **Subcritical-supercritical-subcritical transition**

To demonstrate the capability of the model, flow regime transition is simulated using CCHE2D. The channel of the flow has a contraction reach in which the width of the channel has been greatly reduced (5:1). Both upstream and downstream boundary conditions are subcritical, the super critical flow occurs in the reach when the flow is getting into the downstream narrow channel. Affected by the downstream subcritical boundary condition, the super critical flow abruptly changes to subcritical through a hydraulic jump (Figure 4.9). In this simulation the bed is set to be smooth with a slope of 0.002. The Froude number at inlet, outlet section is 0.1 and 0.89, respectively, and it is 1.12 in the super critical reach. This simulation has not been verified since physical model data is not available.

Figure 4.9 Subcritical-supercritical-subcritical regime transition
Chapter 5

Verification of sediment transport model

Interaction between channel flow and bed materials results in channel changes. In this chapter, bank erosion and change is not concerned, only channel bed elevation changes: aggradation and degradation are simulated and compared with experimental data. Bed change in meandering river is studied qualitatively without comparison due to lack of data.

5.1. Aggradation in alluvial channels

An aggradation flume experiment data (Soni, 1981) was used to verify the sediment transport model adopted. Only bed load transport was observed in the flume experiment. The flume data is shown in Table 5.1.

Table 5.1: Sediment and initial flow conditions (Soni, 1981)

<table>
<thead>
<tr>
<th>q (m²/s)</th>
<th>U (m/s)</th>
<th>d₅₀ (m)</th>
<th>kₛ (m)</th>
<th>H (m)</th>
<th>S_b</th>
<th>F_r</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0355</td>
<td>0.493</td>
<td>0.0003</td>
<td>0.022</td>
<td>0.072</td>
<td>0.00427</td>
<td>0.34</td>
</tr>
</tbody>
</table>

5.1.1. Boundary conditions

- Inlet section: Flow and bed load discharges were prescribed at the inlet section.
- Outlet section: Constant water surface elevation.
• Roughness on the bed: $k_s = 0.022m$ was found to produce better results for the flow, although the aggradation profiles were not sensitive to this value.
• Wall boundary: Total slip condition for flow, and zero normal gradient condition for bed load and bed change increments.

Figure 5.1 shows the comparison of the simulation and the data. The trend of the channel bed parallel increasing of with time is clear and the agreement with the experimental data is good.

![Aggradation bed elevation profiles](image)

Figure 5.1 Verification of sediment transport model using aggradation experiments.
5.2. Degradation in alluvial channels

Physical experiment for studying the channel degradation conducted by Newton (1951) was simulated by using the CCHE2D model. The effective bed roughness of the case is evaluated by van Rijn's method (1989) and the non-equilibrium sediment transport relationship (Bell and Sutherland 1983) is also applied. More details can be found from the M.S. thesis (Chaudhry, 1996). The flume data is listed as followings:

Table 5.2: Sediment and initial flow conditions (Newton, 1951)

<table>
<thead>
<tr>
<th>q (m³/s)</th>
<th>U (m/s)</th>
<th>d₅₀ (m)</th>
<th>kₛ (m)</th>
<th>Sₛ</th>
<th>Fᵣ</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0185</td>
<td>0.45</td>
<td>0.00069</td>
<td>0.041</td>
<td>0.00416</td>
<td>0.50</td>
</tr>
</tbody>
</table>

5.2.1. Boundary conditions

- Inlet section: Flow and bed load discharges were prescribed at the inlet section. In this degradation case inlet bed load was zero.
- Outlet section: Constant water surface elevation.
- Roughness on the bed: kₛ value used was calculated from van Rijn's formula.
- Wall boundary: Total slip condition for flow, and zero normal gradient condition for bed load and bed change increments.

Figure 5.2 demonstrated the simulated and measured degradation processes. The trend of decreasing of the channel bed elevation agrees with those observed in physical experiments. It has been found that estimation of the roughness height due to bed form created in the erosion process is very important to simulation of degradation (Chaudhry, 1996), and van Rijn's (1993) formula produced the most satisfactory results.
5.3. Bed morphological change in meandering rivers

In meandering rivers, the water flows along curved channels, the generated secondary flow may change the bed load moving the direction from that of the main flow, and thus may strongly affect the formation of bars and bank erosion. Figure 5.3 and 5.4 shows the process of bed morphological change of a curved (sine-generated) channel with the initial flat bed, which is described by equation (3.1) with $\theta_e = 110^\circ$. Englund's (1974) relationship of estimating the angle between the direction of sediment motion affected by secondary flow and that of the main flow was adopted for this simulation.

At the early stage of the simulation (Fig. 5.3), bars and pools developed along the convex and concave banks, respectively, and the largest change occurred near the outlet of the channel and these morphological changes gradually developed towards upstream. Figure 5.4 shows longer time simulation results: bar and pool formations become more and more
enhanced, and the upstream bed forms were less developed than those of downstream. Another distinctive feature is that bars and pools were not symmetric to the apex of the meander bends, being shifted to downstream side. As the bars grow higher and pools deeper, dry areas appear on the top of the bars, and the water surface width was reduced except near the cross-overs. These fluvial processes are reasonable behaviors, which can be observed in either flume experiments, and natural rivers.

This verification test only shows the simulation of trend of the bed morphological change in meandering rivers, no comparison was made due to lack of measurement.

Figure 5.3 Simulated bed change in a meandering channel (early stages).
Figure 5.3
Simulated bed change in a meandering channel.
5.4. **Simulation of alluvial river channel migration**

Channel migration is the direct result of asymmetric bed erosion in curved channels due to secondary current. The simulations demonstrated above have shown how much the secondary current can direct bed load sediment motion and change the channel bed topography. If one assumes the channel banks are also erodable, migration would occur and the pattern of the river would change from a mild curved channel to a complex meandering loop system.

Figure 5.5 Simulated channel migration. The channel is of laboratory scale.
It is recognized that the bank erosion involves two processes, frictional erosion and mass failure. The former is caused directly by the shear stress on the bank surface, and the later is due to mechanic failure of bank material triggered by bank toe (basal) erosion, which is also determined by shear stress. Bank failure occur only if the toe erosion exceeded certain limit and the exact time of failure is related to many hydrological processes such as pore water pressure, water table, vegetation density and flood wave, etc. CCHE2D bank model could incorporate all of these processes and simulate channel migration according to local flow condition and bank material properties. Figure 5.4 demonstrates the simulation of an idealized example of channel migration. The channel is of laboratory flume size, uniform sediment for both bank and bed is used and the flow condition at the upstream boundary was steady state.

It can be seen that both of the expansion and downstream shift of the meander channel bend were simulated: curved bends become wider and wider and migrate to downstream at the same time. Eroded bank material were transported to downstream and accumulated on the point bar of the opposite banks. If one continues, the channel bends would grow more and more curved and cut off would eventually occur. More details of this study can be found in the Ph. D. dissertation of Duan (1999).

5.5. Suspended sediment transport simulation in River Nechar, Germany

The numerical model CCHE2D has been applied to simulate flow field for the Lauffen Reservoir on the Neckar River, Germany (Xu, et al, 2001). The model predicted water surfaces and velocity fields under unsteady states. The simulation results showed that predicted water surface elevations were in a good agreement with the field measurements and imply that the CCHE2D model can simulate a unsteady natural river channel flow with a complicated geometry with sharp bends and wider flood plains.
The Lauffen Reservoir on the Neckar River is a channel reservoir, built in 1938. The Lauffen Reservoir was located near the Stuttgart, Germany. It stretches 12 km long from the upstream Bitigheim to the downstream Lauffen with two 180° sharp bends and wide large floodplains in the reach between 129.0 km and 132.3 km. Since 1950 the suspended sediment inflow of the reservoir decreased considerably due to the construction of a cascade of 13 dams in the upstream channel. According to channel profiles in the Lauffen Reservoir surveyed by Wasser- und Schifffahrtsverwaltung, Germany, in different years, the channel bed elevations have been changed considerably, especially in the lower reach between 125.2 and 126.0 km. Over the period of 1950 to 1973, sediments of about 2 m in the depth were deposited in the area near the weir structure between 125.2 km and 130.0 km. The discharge at the gauge station of Lauffen varies from 14.1 m$^3$/s to 1650 m$^3$/s. The mean discharge is 88.5 m$^3$/s. The Enz River discharge into the Neckar River, downstream of the Bitigheim Station. The mean discharge of the Enz River is 20 m$^3$/s.

Numerical simulations were carried out to validate CCHE2D model’s capability to simulate suspended sediment transport in natural rivers. Figure 5.6 shows the whole simulated reach of Neckar River, the blue color on two sides of the channel with zero sediment concentration actually represents bank areas higher than water surface level. The width of the channel is controlled automatically by the moving boundary method of the model. More detailed discussion on this capability can be seen in Chapter 6. The color in the channel represents the suspended sediment concentration in the river. The boundary condition of the sediment is measured at Bitigheim, the concentration is higher on the right bank. The difference of the concentration between the left and right bank decreases gradually towards to downstream due to literal mixing. One of the purposes of the simulation is to evaluate the capability of the CCHE2D model for adequately computing suspended sediment transport processes.

Figure 5.7 shows the comparisons of the measured and simulated sediment concentration in several cross-sections along the river form 136km (upstream) to 128km (downstream). The sediment concentration distribution in both literal direction and longitudinal direction are predicted very well.
Figure 5.6 Simulated suspended sediment concentration in Neckar,
Figure 5.7  Comparison of simulated and measured suspended sediment concentration in several cross sections in Neckar River.
Chapter 6

Feasibility studies

6.1. Flow around spur dikes in natural channel

The spur dike is one of the in-stream structures that is constructed in natural rivers or streams to protect the banks from erosion. The flow pattern, turbulence and vortices generated by the spur dikes are important not only to bank protection and local scouring processes, but also to the local fish habitat. Figure 6.1 shows the simulated flow pattern in the Hotophia Creek, a natural stream in Northern Mississippi. Flow discharge, the size and locations of the spurs are hypothetical. Some portions of the bed with high elevation (include the two spurs) are shown by dry areas. Color shading in the channel represents water surface elevation, which is high at where the flow is blocked. Vectors show clearly the flow pattern in this reach and recirculations behind the dikes are predicted.

Figure 6.1  Simulated flow field around spur dikes in Hotophia Creek.
The complicated bed topography is shown in Fig. 6.2. The color shading represents the bed elevation. Spur dikes and some portion of the banks is higher than the water surface, some gray curves distinguish the water edges. The moving boundary technique was applied to handle location of the water edges with satisfactory results.

![Bed topography of the simulated reach of Hotophia Creek.](image)

**Figure 6.2** Bed topography of the simulated reach of Hotophia Creek.

### 6.2. Unsteady flow in a natural river channel

To demonstrate the model's capability to simulate unsteady flow in highly irregular channel topography, a simulation of flood routing in Hotophia Creek, a natural river in north Mississippi, has been conducted. At the inlet boundary, a sharp hydrograph was prescribed, and the downstream water surface elevation at the outlet boundary was set free. The attached figures (Figure 6.2) show the velocity and water surface at different times in the flood period. Initially dry areas of flood plains were submerged under the flooding and became dry again in the retreat period of the flood.
Simulation of Hotophia Creek during a hydrograph

**Hydrograph parameters**

\[ Q_b = 3 \text{ m}^3\text{s}^{-1} \], \[ Q_p = 200 \text{ m}^3\text{s}^{-1} \]
\[ m = 5 \], \[ T_p = 800 \text{s} \]

**Base Flow:** \[ Q = 2 \text{ m}^3\text{s}^{-1} \]

**Hydrograph:** \[ T = 200 \text{s} \]

**Hydrograph:** \[ T = 500 \text{s} \]

**Hydrograph:** \[ T = 800 \text{s} \]

---

Figure 6.3a  Simulated unsteady flow in Hotophia Creek (rising stage).
Figure 6.3b  Simulated unsteady flow in Hotophia Creek (falling stage).
6.3. **Unsteady Flow Simulation in River Nechar, Germany**

The numerical model CCHE2D has been applied to simulate flow field for the Lauffen Reservoir on the Neckar River, Germany (Xu, 1998, Xu, et al, 2001). The model predicted water surfaces and velocity fields under unsteady states. Manning’s coefficient was identified in the model calibration using measured field data. The calibrated model was then validated using more filed data measured during several flood events. The results showed that predicted water surface elevations were in a good agreement with the field measurements and imply that the CCHE2D model can simulate a unsteady natural river channel flow with a complicated geometry with sharp bends and wider flood plains.

The Lauffen Reservoir on the Neckar River is a channel reservoir, built in 1938. The Lauffen Reservoir was located near the Stuttgart, Germany. It stretches 12 km long from the upstream Bitigheim to the downstream Lauffen with two 180° sharp bends and wide large floodplains in the reach between 129.0 km and 132.3 km. Since 1950 the suspended sediment inflow of the reservoir decreased considerably due to the construction of a cascade of 13 dams in the upstream channel. According to channel profiles in the Lauffen Reservoir surveyed by Wasser- und Schifffahrtsverwaltung, Germany, in different years, the channel bed elevations have been changed considerably, especially in the lower reach between 125.2 and 126.0 km. Over the period of 1950 to 1973, sediments of about 2 m in the depth were deposited in the area near the weir structure between 125.2 km and 130.0 km. The discharge at the gauge station of Lauffen varies from 14.1 m$^3$/s to 1650 m$^3$/s. The mean discharge is 88.5 m$^3$/s. The Enz River discharge into the Neckar River, downstream of the Bitigheim Station. The mean discharge of the Enz River is 20 m$^3$/s.

To demonstrate the model’s capability of the computing unsteady flow with irregular channel morphology, simulated flow field at the peak flood discharge of 1644 m$^3$/s (flood 1990) in the channel bends is presented in Figure 6.4. The area colored by blue in this figure shows the bank areas higher than flood stage. One can see clearly the flow
patterns on the floodplains, secondary circulation driven by the main channel flow, near the station of 130 km and 131.2 km, and island zone, from 130.55 km to 130.8 km. The flow velocities near the inner banks are larger than that those near the outer banks in the main channel.

Figure 6.4 Simulated flow pattern in the Neckar River. Moving boundary technique makes the unsteady flow simulation possible. Detailed flow circulation on the sallow areas driven by the main-channel flow can be observed.

To evaluate the quality of calibration the CCHE2D model was then applied to simulate three flood events (1978, 1990, 1993) with the calibrated Manning’s coefficients. The water surface elevations recorded at the gauge station Bitigheim during these flood events were used for the model validation. The hydrograph of these floods have different
characteristics, different flood duration, total volumes, peak discharges and discharge varying rates. The highest peak discharge was 1644 m$^3$/s in the flood 1990, which is close to the highest historical discharge for the study reach. The validation using this data set would enhance our confidence on the model’s applicability to simulate unsteady flow in irregular natural channels.

Figure 6.5 shows the comparison of the predicted unsteady water stages and measured data at Bitigheim station. The agreements of the comparison are satisfactory. The average differences between predicted and measured water stage at the Bitigheim station are 0.17, 0.13 and 0.15 m for the floods of 1978, 1990 and 1993. The maximum difference is 0.27m at the discharge of about 958 m$^3$/s in 1993, the hydrograph near this point has steep gradient.

Figure 6.5   Comparison of the simulated and measure water surface elevation for three floods in River Neckar, Germany.
Figure 6.6 shows the comparison of simulated and measured free surface elevation along both bank lines of the Neckar River. The hydrograph of the flood event was used as boundary condition. A fixed water stage of 169.7 m at the downstream boundary was applied, because there is a reservoir and the flow stage change little during floods. Bed material in the study reach is non-uniform and varies from coarse gravel to sand and fine clay from upstream to downstream. The Manning’s coefficient calibrated varies from 0.017 to 0.031 in the main channel from downstream to upstream and from 0.04 to 0.06 on the flood plains. Field data were measured along the river course near the left and right banks at the peak discharge of 1644 m$^3$/s of flood 1990. Figure 6.6 shows the reproduced water surface elevation compared with measured data. From this figure one can see that the general agreement between the computation and measurements is good. The average difference between observed and predicted water surface elevation, $\Delta h$, along the channel is less than 0.17 m, and the relative error, $\frac{\sum \Delta h_i}{d_i}$, is smaller than 4.6 %, where d is the water depths at the measured position, and i is a number of measured data. There exist larger differences in water surfaces near the inner and outer banks in the sharp bend reaches. The difference of water surface elevation at the left and right banks at the station 130.2 km is 8 cm, 171.95-171.87 m (compared with measured data, 3 cm, 171.73-171.76) and at the station of 130.8 km, 26 cm, 172.52-172.26 m (compared with measured data, 26 cm, 172.61-172.35 m). Because the actual bank shape and roughness strongly affect the local water surface elevation and velocity, and these information are not available to the simulation, one should therefore expect certain errors along the bank lines.

Figure 6.6  Comparison of simulated and measured surface elevation in a flood event along the channel of River Nechar, Germany.
6.4 Simulation of flow in East Fork River

The East Fork River lies in the Wind River Range of Wyoming west of the Continental Divide and east and south of Mt. Bonneville. The study reach is about 3.3 km in length and with the width ranging from 16 m to 42 m. The geometry of this reach is shown in Figure 6.7a, which was measured in 1979.

The study reach of the East Fork River is typically meandering. The sediment composition of the riverbed is predominantly sand. There’re point bars at each bend and an island near the downstream end where the river bifurcates first and then confluents. The high flow season was caused by spring snowmelt in the mountains. The high flow discharge may be 10 times as much as that of the low flow. Diurnal fluctuations are characterized by a rising stage during the morning, a peak stage at midday, and a falling stage during the afternoon. Flow measurements were made in spring of 1979 at 39 cross sections along the study reach.

CCHE2D was used to simulate flow conditions in this meandering river reach. The simulation domain is discretized by $35 \times 480$ meshes for very low flow and $33 \times 480$ for others. At low discharge, the river width is not full. In this case, moving boundary technique is used and the Manning’s roughness coefficient, 0.02 for low flow and 0.025 for high flow, was calibrated by using the surveyed surface elevation profile.

The flow details for a low flow rate and stage in the river bends (Fig. 6.7b) and near the bifurcation at the island (Fig. 6.7c, d) are simulated very well. At the high flow rate and stage, Point bar in the bendway is submerged (Fig. 6.8a) and flow circulation occurs near the inner bank of the river bend (Fig. 6.8b, c). The moving boundary method makes the unsteady flow simulation in natural channel a relatively easy task.
Figure 6.7. East Fork River simulation reach (a). At low flow rate, the flow stage is low and banks and point bar of a sharp bend are emerging (b). An island virtually becomes bank (c) and (d).
Figure 6.8 At higher flow rate and stage, the point bar and bank are submerged (a). At downstream, a large portion of the island is submerged, one can see the bifurcation of the flow due to the island (b) and (c).
6.5 Simulation of flows associated with in-stream hydraulic structures

6.5.1. Red River Lock & Dam structure

To demonstrate the model’s capability to simulate flows in highly irregular domain and complex conditions as those of natural channels with hydraulic structures, the simulation of flow in a reach of Red River, in Louisiana is shown in Figure 6.7. This reach is just down-stream of the Lock&Dam No. 2. Guide walls were constructed near the Lock for barges to pass the navigation channel, three submerged spur dikes were built on the right bank to train the river flow for the navigation safety and reduce sedimentation. The river bed has highly irregular topography caused by the sedimentation processes induced by flow due to these structures.

The CCHE2D model has been applied to simulate the flow in this reach with different flow discharges. Figure 6.7 depicts the flow pattern at the immediate downstream of the Lock: a big eddy is formed near the downstream entrance of the Lock, which is generated by the shear flow between the main and navigation channels. Smaller scale eddies are also present in the navigation channel. The maximum strength of the big eddy appears a little away from the guide wall. This is reasonable because there is a submerged dike at the tip of the riverside guide wall which restrict the local water depth and therefore reduce the total shear force between the main flow and that in the navigation channel. The computational results are used to help the US Army Corps of Engineers to improve navigation operation, sedimentation dredge, and engineering design.
Figure 6.7  Simulated flow pattern at downstream of Lock&Dam 2, Red River

6.5.2. Mississippi River, Old River Complex

Old River Complex is located at the lower Mississippi River, Louisiana. Three channels were built to discharge the flow from the Mississippi to the Affalachia River. These channels are operated under different flow conditions for different purposes. The channel to the hydro-power plant is often opened for electronic power generation. Other channels are mainly for the flood control.

CCHE2D has been applied to simulate the flow in the Mississippi as well as in these channels. To test the model’s capability in simulating flows in large-scale natural river channels is the main objective. Because the model has moving boundary capability, flood plains, islands are exposed due to the low water stage. Many zones of dead water, separated from the main channel flow were formed when the flow stage descends from a higher level. The main channel is naturally formed along the lower part of the
computational domain, by passing the island and flows into the discharge channels. Figure 6.9 shows the flow details near the inlet of the channel to the hydro-power plant. The water stage in the figure is higher than that in Fig 6.8 because of the higher flow discharge. As a result, more flood plain is submerged under slow moving flow. There is a small island at the junction of the main channel and intake channel, it is submerged by the high flood but the flow over the top is still affected the island.
Figure 6.8   Simulation of channel flow in Mississippi, Old River Complex.
6.5.3 Simulation of flow with spur dikes and submerged weirs in Victoria Bendway, Mississippi.

Because numerical modeling is more efficient and cost-effective than physical modeling, NCCHE was contracted by the ERDC (Engineering Research & Development Center) of the US army Corps of Engineers to investigate the effect of submerged weirs on the navigation condition in the Victoria Bendway of Mississippi River. Numerical models are to be used to simulate the flow field with and without the submerged weirs under the same flow condition. Comparisons of the simulation results would reveal how and why these weirs affect the flow field and thus navigation. Victoria Bendway was selected as the study site, not only because it has six submerged weirs installed but also because it was a study site where a substantial amount of data (bed elevation, velocity, stage, etc.)
had been collected, which provide excellent data sets for numerical model verification and validation. As part of this project, the two-dimensional model solution of the flow field was compared with the depth-integrated measured data obtained from the field. This comparison was intended to check the consistency of the numerical solutions and the measured data.

Figure 6.10 shows the cross-sections for measuring velocity field in the Victoria Bendway. At each survey point the depth-averaged velocity is shown, the lines that the velocity vectors are drawn are actually the survey paths. The data were taken in June, 11, 1998 and June, 12, 1998. The velocity data measured in June, 11, 1998 has 17 sections with total 2210 points while the data taken in June, 12, 1998 includes 17 sections with total 2494 points, those sections having problems in the survey process are excluded. The data taken in June, 11, 1998 were used for the 2D comparison. It should be noted the section numbers (from 1 to 34) for these measurement used here are not the same as the survey line number which is difficult to be used. The conversion of the section number and the survey line number is listed also in Fig. 6.10. Linear interpolation (inverse distance) was used to interpolate the computed solutions at the four vertices of a mesh cell in which the data point is contained.
Figure 6.10  Victoria Bendway and the survey lines, depth-averaged velocities are shown.
Figure 6.11  Comparison of the computed depth-averaged velocity (m/s) and that of measured (June, 11, 1998). Black points are velocities, red points are reported errors in the measured data.

Figure 6.11 shows the comparison between the depth-averaged two-dimensional prediction and the depth-averaged velocity data. The general agreement of the depth-averaged velocity is clear. The mean value of measured velocity is about 1.5 m/s while that of the computation is about 1.4m/s. Velocities higher than 1.5m/s have larger errors than those of lower ones. Discrepancies between the computed results and measurements and data scatters may be due to the following reasons:

- Errors in the measurements (±0.2m/s, Fig 8.4),
- Errors due to depth-averaged model approximation
- Error created in interpolation,
- Bed elevation data used in computation are different from those during field data collection,
- Errors caused by the adoption of a constant Manning’s $n$ in simulations of the flow field.

These factors would cause error in velocity distribution in the cross-sections as well as the velocity directions at each point. Nevertheless the overall agreement is still very good considering the only calibration performed was in the selection of an appropriate Manning’s coefficient, an averaged value being used in the entire bendway during simulations.

Figure 6.12 shows the simulated flow field by CCHE2D including the effect of submerged weirs. One can observe that the acceleration and deceleration of the flow in the main channel due to the submerged weirs and the flow pattern over the point bar as affected by the spur dikes. There are three spur dikes on the point bar, all of them are submerged due to high flood stage of the flow condition, only a small portion of the two higher dikes are exposed. The first dike plays an important role to direct the flow into the main channel, only small amount of flow pass over the top of dike because the water depth there is every little. The height of the third dike is low, the flow depth over it is quite large, the blockage to the flow is less significant than the other two dikes.
Figure 6.12  Velocity magnitude (m/s) and vectors in the Victoria Bendway without submerged weirs, CCHE2D
6.5.4 Simulation of flow in the channel downstream of Wanan Reservoir-A case with dike field

Wanan Reservoir was constructed on Wanan River, China. Since it was in operation in 1970, strong scour appeared in the downstream channel. To maintain reasonable condition for navigation and protect the banks from strong erosion, many spur dikes were constructed attaching to both banks with difference angles. This case has been simulated to test CCHE2D model’s capability of reproducing the flow field in a channel reach of 25km long and 600m wide with multiple dikes. Because the orientations of the dikes are not perpendicular to the main channel, some of them have sharp angle to the bank, the mesh generated for this case was highly irregular. The bed bathymetry was complicated, the main channel part was quite deep and some part of the channel between dikes and islands are emerged from the water surface from time to time which has to be modeled using moving boundary technique.

Simulated Flow Field in A Channel With Numerous Dikes Protecting the Banks

Figure 6.13. Simulated flow field in Wanan River with multiple dikes protecting banks
Figure 6.13 shows detailed flow of a small portion of the simulated channel. The red color indicates the dikes and the rest color shading represents the bed elevation. The blue color represents deeper channel and the green shows higher bed. One can see somewhere the bed is higher than the water surface and they were not covered by the vectors. The flow discharge for this case was 1000m$^3$/s. The maximum flow velocity was about 2.5m/s. Unsteady flow and sediment simulation for this case shall be reported later.

Figure 6.14 Simulated flow in Wanan River close to the Dam of Wanan Reservoir.
Chapter 7
Conclusion

The CCHE2D is a depth integrated two dimensional model developed for studying unsteady, turbulent, free surface open channel flows and sediment transport problems in channels with highly irregular topography and bank protection structures. Comparisons of the simulated results with analytical solutions (parabolic flow, pollutant transport), physical model data (curved channel flows, hydraulic structures, compound channel flows, super critical flows, trans-critical flows, channel aggradation, degradation), and field data (curved channel with structures, unsteady flow, suspended sediment transport) have shown good agreements. The predictions are quire realistic when the model is applied to simulate some hypothetical cases such as channel erosion, migration, and flow patterns in natural rivers.

Based on our experience, the CCHE2D model is efficient and capable of predicting realistic flows and sediment transport in streams. It is a useful tool for hydraulic researchers to study open channel flows and sediment transport problems, and for engineers to design hydraulic structures and plan hydraulic engineering projects.

The authors are the first ones to point out that the state of the art in modeling flow and sedimentation processes in alluvial rivers is being continuously advanced at the present. The NCCHE plans to release new versions of CCHE2D with a higher level of capability as soon as they are developed and verified.
REFERENCES


De Vriend, D.J., Flow measurements in a Curved Rectangular Channel, Laboratory of Fluid Mechanics, Department of Civil Engineering, Delft University of Technology, Internal report no. 9-79.


Xie, B.L. 1994, “Flow data measured in a channel with sudden expansion”, Wuhan University, China, Private communication.

