3D Numerical Simulation of Supercritical Flow in Bends of Channel


Abstract—An attempt has been made to simulate supercritical flow in curved open channels using three-dimensional CFD analysis. The CFD analysis is carried out on two 45° curved open channels using the commercial software package FLUENT. The volume of fluid (VOF) model is used to simulate the air-water interaction at the free surface and the turbulence closure was obtained using the standard $k-\varepsilon$ turbulence model. Verification of the model was carried out by comparison of FLUENT results and experimental data in a one side contraction. Finally, the results of numerical model are compared with available experimental data of supercritical flow in channel bends. Results of the study clearly demonstrate the strengths of using CFD as an alternative or compliment to physical modeling.

Keywords—Bend, 3D Simulation, Supercritical Flow, VOF Method.

I. INTRODUCTION

The investigation presented in the following is on the behavior of flow at high velocities in curved open channels. High velocity flows introduce problems in hydraulic engineering which cannot be solved easily. Chief among these problems is the formation of standing waves, which are usually unpredictable and require expensive high-walled structures to prevent overtopping [5].

Methods of design used in the analysis of low velocity structures have been found insufficient and cannot be utilized to predict standing wave formation [2]. Many high velocity channels have of course been designed, and with some degree of success. But these channels were designed by empirical methods without any contribution to our understanding of the fundamental factors which underlie the problem. Any obstruction to the path of flow generates a surface wave that extends across the flow domain and is carried downstream. The result is an oblique standing wave, the height of which could be considerably larger than the normal depth of flow.

Predicting the possible locations of oblique standing waves and determining the elevations of the water surface is necessary to design the required wall heights to avoid overtopping [7].

The prediction is extremely difficult because the flow in open channel is usually transient and turbulent, the geometry is irregular and curved, and the free-surface elevation is varying with time and space. The free-surface variations would change the hydrostatic component of water pressure and depth of water [1]. To date, predictions of flow in open channel are mainly focused on one-(1D) and two-dimensional (2D) simulations, in which the influences of free-surface variation are usually ignored. Therefore, to properly approximate the reality, the supercritical flow in channel bends should be considered and simulated by three-dimensional (3D) CFD model.

The past analytical studies of supercritical flow in channel bends are completely reviewed by Ghaeini-Hessaroeyeh and Tahershamsi (2009). Recently, some researchers have attempted to solve this problem with a method of numerical models. The computational approach to channel bends under supercritical flow was introduced by Ellis and Pender (1982). Dammuller et al. (1989) presented a two-dimensional mathematical model for studying unsteady flows in curved channels. Elliot and Chaudhry (1992) extended the traditional characteristic methods for wave propagation to two-dimensional finite element model for high-velocity channels. Valiani and Caleffi (2005) carried out the analysis of shallow water equations suitability to numerically simulate supercritical flow in sharp bends. Ye et al. (2006) investigated S-shaped spillway using physical and numerical models. Yu-chuan and Dong (2007) simulated dam-break flows in curved boundaries by finite-difference method, in a channel-fitted orthogonal curvilinear coordinate system [6].

Most computational analysis of curved open channel flows conventionally adopt averaged De St. Venant equations. These equations are based on the fundamental assumptions of uniform velocity profile and hydrostatic pressure distribution. In this memory the results of the simulations in curved open channels are presented, without holding the usual hypothesis of hydrostatic pressure. The computed water surface profiles of two bends are compared with available experimental data obtained in laboratory. Excellent agreement is found in the comparisons.
II. NUMERICAL SIMULATION

A. Governing Equations

Numerical modeling involves the solution of the Navier-Stokes equations, which are based on the assumptions of conservation of momentum and mass within a moving fluid. The $k-\varepsilon$ turbulence model (Launder and Spalding, 1972) was used for its extensive application in practice. The continuity equation, momentum equation and equations of $k$ and $\varepsilon$ are given as follows [4]:

Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$  \hspace{1cm} (1)

Momentum equation

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$  \hspace{1cm} (2)

$k$ Equation

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu \frac{\partial k}{\partial x_i} \right] + G - \rho \varepsilon$$  \hspace{1cm} (3)

$\varepsilon$ Equation

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\mu} \frac{k^2}{\varepsilon} - C_{\varepsilon} G$$  \hspace{1cm} (4)

Where $\rho$ = volume fraction average density; $t$ = time; $u_i$ = velocity component in $x_i$ coordinate ($i = 1,2,3$); $\mu$ = volume fraction average molecular viscosity; $p$ = modified pressure; $C_{\mu} = 1.44$; $C_{\varepsilon} = 1.92$ (constants of the $\varepsilon$ equation); and $G$ = generation of turbulent kinetic energy which can be given as

$$G = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_i}$$  \hspace{1cm} (5)

Where $\mu$ = turbulent viscosity and it can be calculated as

$$\mu = \rho C_{\mu} \frac{k^2}{\varepsilon}$$  \hspace{1cm} (6)

Where $C_{\mu} = 0.09$ (experimental constant); $\sigma_k = 1$ (turbulence Prandtl numbers of $k$); and $\sigma_{\varepsilon} = 1.33$ (turbulence Prandtl numbers of $\varepsilon$).

B. The Free-Surface Treatment

The flow is modeled in FLUENT using volume of fluid (VOF) formulations. The VOF method was proposed by Nicholas and Hirt (1975) [8]. The flow involves existence of a free surface between the flowing fluid and the atmospheric air above it. The flow is generally governed by the forces of gravity and inertia. In VOF model, a single set of momentum equations is solved for two or more immiscible fluids by tracking the volume fraction of each of the fluids throughout the domain. The mathematical formulation adopted is described briefly below.

The basic idea of the VOF method is to define and to represent the volume fractions of water and air, respectively, in the computational domain, and the sum of fractions of water and air is 1 in each computational cell.

$$\alpha_w + \alpha_a = 1$$  \hspace{1cm} (7)

The differential equation of volume fraction of water is as follows:

$$\frac{\partial \alpha_w}{\partial t} + u_i \frac{\partial \alpha_w}{\partial x_i} = 0$$  \hspace{1cm} (8)

By solving the above continuity equation (8), the interface between water and air can be determined.

C. Mesh Generation

The Gambit software was used to create and mesh the geometry. A structured mesh was used to discretize the geometry for simulation. This mesh was comprised of hexahedral elements. The mesh size used in both cases was $1*5*5$ cm in $z$, $y$ and $x$ directions, respectively. The final meshes used had the following node numbers $26*5*246$ in both channel bends cases. For grid independency the numbers of nodes were doubled in all three directions. The absolute mean differences were calculated for maximum water depths. The largest difference which occurred in any depth by doubling the node numbers was small enough that for all practical purposes the results could be termed as grid independent. A post processing check on mesh quality based on assessing the skewness of the generated cells indicated that the mesh is of high quality and would not compromise solution stability.

D. Boundary Conditions

The boundary conditions are discussed referencing Fig. 1. The inlet boundary included a water inlet and air inlet. The velocity inlet boundary was adopted as the water inlet and pressure inlet was adopted as the air inlet boundary. Side and bottom walls were modeled as no-slip wall boundary condition. A pressure outlet boundary condition is used at the channel exit to allow free-air flow. The top of the channel (in contact with air) was set as a symmetry boundary which means that the normal velocity and normal gradients of all variables are zero at this plane of symmetry.

![Fig. 1 Boundary Conditions](image)

E. Model Options

Numerous options are available in the FLUENT software package to model the flow field. It is beyond the scope of this paper to discuss all of the options available in FLUENT; however, relevant aspects of the model are presented.
**FLUENT** offers the user a choice of the numerical scheme used in the solution of the governing equations. A pressure based solver was selected whereby the non-linear governing equations are solved sequentially in an iterative process. The governing equations are linearized implicitly. A first order upwind solution was selected for momentum equations, turbulent kinetic energy, and turbulent dissipation rate. The Geometric Reconstruction scheme was used for the volume of fraction and the PISO scheme was selected for pressure-velocity coupling. Finally, the PRESTO scheme was used for pressure interpolation as recommended by **FLUENT** for multi-phase flows [9]. The multi-phase flow computations were handled using an explicit VOF approach.

The iterative nature of the solution schemes employed requires that convergence criteria be specified for terminating the iterations. A commonly used procedure to identify a converged solution is to use a residual definition. This definition implies that convergence is reached when the normalized changes in variables between successive iterations are equal to or less than a criteria limit. In these cases the limits were set as 0.00001. The calculation was initiated using the default under-relaxation coefficients provided in **FLUENT**. A proper time step, which provides converged results, is a function of grid size applied to the problem. For these two cases, considering $\Delta t = 0.001 \text{s}$ and $v_{\text{fluid}} = 3$ and $3.272 \text{ m/s}$, the maximum value of the Courant numbers over the domain become $\text{Courant number} = v_{\text{fluid}} \Delta t / \Delta x = 0.03$ and $0.03272$, respectively. So, this time step is sufficiently small to provide convergence at every time step during the entire simulation.

### III. VALIDATION OF CFD MODEL

For validation of CFD simulation, the contours of water surface obtained from CFD model in a one side contraction is compared with the experimental data. The upstream Froude number equals 2.188. The channel’s left wall is vertical and the right wall is tapered. The contraction width is 30 m and 25.27 m at the top and bottom, respectively, with an 8.95° taper angle for right wall.

For CFD simulation, the flow domain is discretized with hexahedral elements of face length equals to 0.1 m, 0.1 m and 0.01 m. The contours of water surface obtained by CFD model and Anastasiou et al. analytical solution are shown in Figs. 3 and 4, respectively. It is seen that they are similar in nature and are in excellent agreement.

### IV. RESULTS AND DISCUSSION

In the mechanism of supercritical bend flow, the beginning of the bend was referred to as the origin of flow disturbance with a positive wave front crossing the channel from the outer to the inner wall, and a negative front crossing the other way. Both waves are reflected and yield to cross waves in the downstream channel (Reinauer and Hager, 1997).
According to Fig. 5 the reflection of wave fronts between the outer and inner bend wall is observed and the flow pattern in the bend is predicted correctly.

A significant experimental study on channel chute bends was conducted by Poggi (1956). The experiments in three bends were conducted, with deflection angles of 30° and 45°, and relative curvatures $B/R = 1/12$ and $1/25$. The bottom slopes were 5 and 10%, respectively. The Froude numbers were between 2.27 and 5.1. The initial conditions of the selected test data of Poggi are shown in Table I.

| TABLE I |
| Units for Magnetic Properties |

<table>
<thead>
<tr>
<th>Channel/Case</th>
<th>Test</th>
<th>Dev. Angel</th>
<th>Slope</th>
<th>$B$</th>
<th>$B/R$</th>
<th>$H_o$</th>
<th>$F$</th>
<th>Fig. Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>3</td>
<td>45 (Deg)</td>
<td>5 (%)</td>
<td>0.25 (m)</td>
<td>0.083</td>
<td>9.93 (cm)</td>
<td>3.04</td>
<td>6</td>
</tr>
<tr>
<td>Case 2</td>
<td>5</td>
<td>45 (Deg)</td>
<td>10 (%)</td>
<td>0.25 (m)</td>
<td>0.083</td>
<td>4.68 (cm)</td>
<td>4.83</td>
<td>7</td>
</tr>
</tbody>
</table>

Where $R$ is the axial radius of curvature, $B$ is channel width, $H_o$ is the approach flow depth and $F$ is the Froude number in the beginning of the bend. The results of the present model for the contour plot of water depth in channel 1, test 3 and channel 2, test 5 are shown in Figs. 6 and 7.

According to Reinauer and Hager (1997), the characteristics of the first waves are of interest, because all waves in downstream channel are smaller than the first wave. Therefore, the results of the present model are compared quantitatively in Figs. 6 and 7 to test data of Poggi and a two-dimensional method, WAF method, for the first wave along the outer and inner bend wall, respectively.

A. Case 1

It is observed in Fig. 6 that the results of numerical model are satisfactorily predicting the wave lengths in outer and inner wall and its prediction is much better than the two-dimensional WAF method.

V. CONCLUSION

Vertical motion is always neglected in shallow water equations. This assumption’s effect is apparent in bends cases under supercritical flow and is shown through this study. Due
to this, a fully three-dimensional numerical model has been used to simulate the supercritical flow field in a channel bend. The $k-\varepsilon$ turbulence model with VOF method was used to simulate flow pattern.

The performance of numerical model in handling shock capturing in two cases through comparison with laboratory tests and two-dimensional WAF method were presented in this study. Overall results indicates that the present model has an excellent prediction of flow pattern of supercritical bend flow and also the wave length and wave height in the inner and outer wall.

REFERENCES