Three-Dimensional Flow Model For The Downstream of Kuffa Barrage

Dr. Muhammed J. M. Al-Kizwini*, Dr. Saleh I. Khassaf**
& Majid H. Hobi**
Received on: 25/2/2010
Accepted on: 5/1/2011

Abstract

The three-dimensional numerical computational fluid dynamics “CFD” computer program "SSIIM" was used to predict the flow field downstream the Kuffa Barrage. It solved the Reynolds-Averaged Navier–Stokes equations in three dimensions to compute the water flow and used the finite-volume method as the discretization scheme. The model was based on a three dimensional, non-orthogonal, structured grid with a non-staggered variable placement. The comparison between filed measurements and numerical results were considered to make the correct decision in this model. The results showed that the maximum velocities were inclined from the river center. The determination coefficients for distribution of velocities ranged from 0.94 to 0.96.

Keywords: Three dimensions, CFD, SSIIM, Kuffa barrage, Reynolds-Averaged Navier–Stokes

Introduction

The Navier-Stokes equations for turbulent flow in a general three-dimensional geometry are solved to obtain the water velocity. The k-ε model is used for calculating the turbulent shear stress. A simpler turbulence model can be used. This is specified on the function data in the code of Model (F 24) in the control file of SSIIM program.

The Navier-Stokes equations for non-compressible and constant density flow can be modeled as:

\[
\frac{D\overline{u}_i}{Dt} = -\frac{1}{
\rho \frac{D\overline{u}_i}{Dx_j}} \frac{\partial P}{\partial x_j} + \frac{1}{
\rho \frac{D\overline{u}_i}{Dx_j}} \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \overline{u}_i}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \overline{u}_i}{\partial x_j} \right) \tag{1}
\]

The left term on the left side of the equation is the transient term. The next term is the convective term. The

* Building and Construction Engineering Department, University of Technology /Baghdad
** Engineering College, University of Kufa / Kufa

434
The first term on the right-hand side is the pressure term. The second term on the right side of the equation is the Reynolds stress term. To evaluate this term, a turbulence model is required.

The equations are discretized with a control-volume approach. An implicit solver is used, also for the multi-block option. The SIMPLE method is the default method used for pressure-correction.

The SIMPLEC method is invoked by the data set in the control file in the program. The power law scheme or the second-order upwind scheme is used in the discretization of the convective terms. This is determined by the values on the data set in the control file. The numerical methods are further described by Patankar (1980), Melaaen (1992) and Olsen (2000).

The default algorithm in SSIIM neglects the transient term. To include this in the calculations the data set in the control file is used. The time step and number of inner iterations are given on this data set. For transient calculations it is possible to give the water levels and discharges as input time series, Zhou Liu, (2001).

2 The turbulent kinetic energy ($k$)–eddy viscosity ($\varepsilon$) model

The $k$–$\varepsilon$ model calculates the eddy-viscosity as:

$$\nu_T = c_p \frac{k}{\varepsilon} \quad \ldots (2)$$

$k$ is turbulent kinetic energy constant, defined by:

$$k = \frac{1}{2} \frac{\partial \rho}{\partial \xi} \quad \ldots (3)$$

$k$ is modeled as:

$$\frac{\partial k}{\partial t} + \frac{\partial (\nu_T k)}{\partial \xi} = \frac{\partial}{\partial \xi} \left( \nu_T \frac{\partial k}{\partial \xi} \right) + P_k - \varepsilon \quad \ldots (4)$$

where $P_k$ is given by:

$$P_k = -\nu_T \left( \frac{\partial u}{\partial t} \right) \left( \frac{\partial u}{\partial \xi} + \frac{\partial v}{\partial \eta} \right) + \frac{1}{2} \left( \frac{\partial u}{\partial t} \right)^2$$

$$\ldots (5)$$

The dissipation of $k$ is denoted $\varepsilon$, and modeled as:

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial (\nu_T \varepsilon)}{\partial \xi} = -c_{\mu} \frac{k^2}{\varepsilon} + c_{\mu} \frac{P_k}{\varepsilon} \quad \ldots (6)$$

In the above equations, the $c$'s are different constants. These cannot be changed by the user.

The $k$–$\varepsilon$ model is the default turbulence model in SSIIM.

3 The kinetic energy ($k$)–Specific dissipation rate ($\omega$) model

In SSIIM, the wall laws for the $k$–$\varepsilon$ model are used also for the $k$–$\omega$ model. This is due to the easier inclusion of roughness.

The $k$–$\omega$ model was developed by Wilcox (2000). It is given by the following equations:

$$\nu_T = \frac{k}{\omega} \quad \ldots (7)$$

$k$ is turbulent kinetic energy, similar to the $k$–$\varepsilon$ model. $k$ is modelled as:

$$\frac{\partial k}{\partial t} + \frac{\partial (\nu_T k)}{\partial \xi} = \frac{\partial}{\partial \xi} \left( \nu_T \frac{\partial k}{\partial \xi} \right) + P_k - \omega \quad \ldots (8)$$

where $P_k$ is the production of turbulence, similar to the $k$-epsilon model:

Instead of using the dissipation of $k$ as the second variable, the model uses $\omega$, which is the specific dissipation rate (units seconds$^{-1}$). The equation for is modeled as:

$$\frac{\partial \omega}{\partial t} + \frac{\partial (\nu_T \omega)}{\partial \xi} = \frac{\partial}{\partial \xi} \left( \nu_T \frac{\partial \omega}{\partial \xi} \right) + \omega \left( \frac{\partial k}{\partial \xi} - \frac{\partial \omega}{\partial \xi} \right) \quad \ldots (9)$$

The following values and formulas are used for the additional parameters.
The velocity-area principal was used to compute discharge from current-meter data. This method was useful to verify the barrage outflow discharges which ranged from 30-155 m$^3$/sec during the study period.

6. Represented Domain in CDF
In computation fluid dynamic (CFD) the prototype domain of flow process has to be described in the numerical model. Initially, the grids of model must be close to the real prototype model for better simulation. The convergence of the prototype regime and the numerical model regime is very difficult and complex in this software. This is because SSIM couldn’t represent natural regime easily. To solve the problem, the SSIM model was connected with other software in order to construct grids. The best solution was to apply a 3-D MAX software. Considering The export and import data files between the two softwares, a technique was used for the first time during this research. The numerical grid consist of 198 rows across the reach with 35 grid cells in each row. Vertically each grid composed of 13 layers, in order to construct geometrically a three dimensional model at the reach, Figures.5 and 6.

7. Velocity Distribution by Numerical Model
The numerical model presents the distributions of velocities for each location in the reach study, Table.2. In addition the model considered many hydraulic variables such as concentrations of sediments distribution, pressures, Froude Number, viscosity, Epsilon, depth, bed elevations, roughness, eddy viscosity, etc.

After calculation was finished,
secondary flow was observed due to slight bends occurred through the regime in the study reach, as shown in Figure 6. The first one at section $i=71$, the second at section $i=133$ and the third one at section $i=170$. Such behavior of flow led the current to deviate from its primary direction (Figures 7 to Figure 9).

The main secondary motion at the water surface is towards the convex bank. Therefore, the flow elements move downwards reaching the bottom. While flow elements are vectored inwards to the concave bank. Consequently, the flow exhibits a cross-circulation. The combination of this cross-circulation and the major flow direction results into fluid spiral motion.

Due to the centrifugal forces acting on the primary flow causing that the fluid element will follow a curvilinear trajectory. This hydraulic phenomena was difficult to observe practically in site. Therefore, experimental verification for the water flow calculation couldn’t be obtained and consequently the numerical model results couldn’t be judged precisely. It was necessary to review the literatures in order to verify the numerical model results for this type of flow.

Figures 7 and 9 exhibits the effect of concavity while Figure 8 exhibits the effect of convexity of the banks on the flow regime. The arrows represent the velocity components $v$ and $w$ in transversal and vertical direction, respectively, and the length of each arrow represent the magnitude of the velocity according to scale in addition to its direction.

In order to present the longitudinal velocity distribution at the chosen cross-sections, the velocities in the vertical layer have been depth averaged, resulting in one representative velocity for each water column. Furthermore, the numerical model uses Cartesian velocity components as dependent variables, so that longitudinal velocity components used in the later comparison must be obtained by transformation to the corresponding directions. Besides the movement of the longitudinal velocity towards downstream also a vertical movement of the maximum velocity was observed. The cross-section at the apexes shows a longitudinal, depth-averaged velocity maximum close to the center of the cross-section. Having a local peak at the close right bank and the beginning of flow toward downstream, as shown in Figure 10A. The Figure shows that the isotaches are compressed, indicating strong velocity gradients. Considering the subsequent cross-sections, the core velocity moves towards the concave bank, while in the vertical direction it drifts towards the center, as can be seen in Figure 10B. In Figure 10C, the lines of equal velocity are stretched out, indicating the core velocity moves toward the concave bank. Same action was repeated in the downstream bends. Many studies observed the same action in bending regimes. Weiming, (2008).

### 8. Model Verification

Verification can be defined as a process for assessing the numerical simulation uncertainty and when conditions permit, estimating the sign and magnitude of the numerical simulation error and the uncertainty in that estimated error. (Figure 12)

However to verify numerical model with prototype the results were
Three-Dimensional Flow Model For The
Downstream of Kuffa Barrage

divided into three parts. The first part
deals with flow calculation at 0.2 of
the depth while the second part deals
with at 0.4 of the depth while the
third part at 0.8 of the depth as
described in Figure.12.

According to the results,
Figures.13 indicates that there is
fairly good agreement between
measured and calculated velocities.
With determination coefficient
ranged from 0.94 to 0.96. One reason
for the deviation between measured
and calculated velocities can be due
to some lack of accuracy in the
measurements of the velocities and to
the geometry of the reach. The
software, estimate the bed form
between the consequent sections
according to the data at these
sections. This will lead to the
geometry to be inexactly modeled.
The largest deviations between the
measured and modeled velocities
were found for velocity distribution
at high velocity values, Figure.12.
This was due to the effect of the
bottom roughness on velocity
distribution “Hydraulically rough
flow”. This phenomena will lead to a
high separation layers from the
bottom to the surface. This was found
to be the main reason behind the
disagreement between the measured.
The other reason for the deviation
between measured and computed
velocities was thought to be the size
of cell in the model. Reducing the
size of the grid cells in areas of small
horizontal distance, will probably
increase the accuracy in these areas.
The decision of number of grid in
each direction must be taken with
experience in numerical modeling.

Velocity calculations by the
SSIIM model at each node were
conducted in three dimensions. The
more node numbers lead the model to
be more time consuming in solving
Navier- Stock’s equation. The grids
are further explained by Olsen
(1999). In a structural three
dimensional grid, each cell will have
three indices, making it easy to
identify grid locations

9. Conclusions

This study presents the
development and comparison
performed in the numerical model
SSIIM and a prototype. The study
examined the model results with
respect to the those observed in the
field in order to determine whether
the numerical model (SSIIM) is able
to predict velocity distribution in the
study reach.

According to the results obtained by
this study, the following points are
concluded:

1. A good relation was observed
between the measured and
computed values of velocity at the
study reach in three dimensions,
with determination coefficients
ranged from 0.94 to 0.96.

2. In the region study a hydraulic
phenomena was observed
(secondary flow). Which effect
the study reach region
hydraulically.

3. The SSIIM is one of the useful
tools to predict the velocity
distributions in three dimensions
which gave good idea about the
behavior of the flow velocities.

Notations
Symbol  Symbol Meaning  Unit

U  average velocity  m/s
ρ  Density of water  m/s
P  Pressure  N/m²
νt  Turbulent eddy-viscosity m²/s
c,C  constants in k-ε turbulence
model

References:
Three-Dimensional Flow Model For The Downstream of Kuffa Barrage


Figure (1) The Map of Region of study

Figure (2) The natural regime of study region, (Google Earth©)

Figure (3) The positions of cross-sections (C.S.).

Figure(4): The current mater that used in his study.

Figure(5 ) Topography of the reach by the model in three dimensional view

Figure (6) Definition sketch of the mesh(Top view)

Figure (7) Velocity vectors plot at cross-section bend no. 1

Figure (8) Velocity vectors plot at cross-section bend no. 2
Three-Dimensional Flow Model For The Downstream of Kuffa Barrage

Figure (9) Velocity vectors plot at cross-section bend no. 3

Figure (10) Contour lines for horizontal velocity distribution A at beginning, C at end of region of study

Figure (11) Horizontal (x-y) velocity

Figure 12: Verification of a numerical model

Figure 13: Determination coefficient for velocity at 0.8d, 0.4d and 0.2d between computed and measured for Q=75.62m³/sec