Exploratory study of the influence of the wake produced by acoustic doppler velocimeter probes on the water velocities within measurement volume

Jorge D. Abad¹, Rodrigo A. Musalem¹, Carlos M. García¹, Mariano I. Cantero¹ and Marcelo H. García²

¹Graduate Research Assistant, Ven Te Chow Hydrosystems Laboratory, Dept. of Civil Engineering, University of Illinois at Urbana-Champaign, 205 N. Mathews Ave., Urbana, IL, 6180, email: abad@uiuc.edu

²Chester and Helen Siess Professor, Ven Te Chow Hydrosystems Laboratory, Dept. of Civil Engineering, University of Illinois at Urbana-Champaign, 205 N. Mathews Ave., Urbana, IL, 6180, email: mhgarcia@uiuc.edu

Abstract

Acoustic doppler technique is widely used in both fields and laboratory facilities to compute the mean water velocity and to characterize the turbulence of a flow. In general they provide the three dimensional components of flow velocity in a measurement volume in the water body with fairly good spatial and temporal resolution for engineering applications. The most sophisticated devices can even gauge a velocity profile measuring the water velocity in several measurement volumes along a line. However, these devices are semi intrusive which might have, depending on the experimental setup, substantial consequences in the measurements obtained due the flow perturbation created by the probe. The goal of this paper is to explore experimentally and numerically the wake effect of the probe on the measurement volume in order to validate the measurements provided by this kind of instruments or incorporate some corrections if needed. A computational fluid dynamic (CFD) model is used to simulate an open channel flow where the model was validated with previous experimental results. In the other hand, the laboratory measurements were conducted in an open channel flume located in the Ven Te Chow Hydrosystems Laboratory of the University of Illinois. The measurements were done using particle image velocimetry technique (PIV) producing two dimensional velocity fields around the acoustic probe measurement volume with and without the presence of the probe. The numerical and experimental ranges of Reynolds numbers (Re) tested were 3\times10⁶ to 1\times10⁷ and 1\times10⁴ to 5\times10⁴ respectively. Non dimensional contour plots showing the difference between the flow velocity and turbulent quantities with and without the probe are built. Both results show that the errors are less than 10 percent around the probe. This methodology is still under development, however it provides more insight for experimental setups and it could be applied to other acoustic doppler instruments such as the ADV (Acoustic Doppler Velocimeter) and ADCP (Acoustic Doppler Current Profiler) among others.
CFD modeling

A commercial code FLOW-3D (Flow Science, 2003) has been used to analyze the flow pattern around the probe. FLOW-3D specializes in the accurate simulation of free surface flows, using the true VOF (Volume of Fluid) technique and it has been used for several purposes.

Flow field and turbulence equations. The mass continuity equation for an incompressible flow is described by equation 1. The momentum equations in x, y and z are described by equations 2 to 4 respectively.

\[
\frac{\partial}{\partial x} (uA_x) + \frac{\partial}{\partial y} (vA_y) + \frac{\partial}{\partial z} (wA_z) = 0.0
\]

\[
\frac{\partial u}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right\} = - \frac{1}{\rho} \frac{\partial P}{\partial x} + G_x + f_x
\]

\[
\frac{\partial v}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right\} = - \frac{1}{\rho} \frac{\partial P}{\partial y} + G_y + f_y
\]

\[
\frac{\partial w}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right\} = - \frac{1}{\rho} \frac{\partial P}{\partial z} + G_z + f_z
\]

Where \( V_F \) is the fractional volume open to flow, \( A_x, A_y, A_z \) are the fractional area open to flow, \( G_x, G_y, G_z \) are the body accelerations, \( f_x, f_y, f_z \) are the viscous terms. Besides the mass and momentum equations, it is required to use a turbulence closure. In this sense, FLOW-3D can handle different turbulence closures such as Prandtl mixing length model, turbulent energy model, \( \kappa - \varepsilon \) model, Renormalized Group (RNG) model and Large-Eddy simulation model. In the present study, the RNG closure has been chosen, because, this model is ideal for situations where high mean strain rate and non equilibrium situations are presented (Yakhot and Nakayama 1986, Yakhot and Smith 1992, Shyy et al. 1997). The RNG model is implemented very similar to the standard \( \kappa - \varepsilon \) described by equations 5 and 6 (turbulent kinetic energy and dissipation of turbulent kinetic energy equations respectively).

\[
\frac{\partial k}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial k}{\partial x} + vA_y \frac{\partial k}{\partial y} + wA_z \frac{\partial k}{\partial z} \right\} = P + \text{Diff} - \varepsilon
\]

\[
\frac{\partial \varepsilon}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial \varepsilon}{\partial x} + vA_y \frac{\partial \varepsilon}{\partial y} + wA_z \frac{\partial \varepsilon}{\partial z} \right\} = \frac{C_{\varepsilon 1} \varepsilon}{k} P + \text{DDif} - C_{\varepsilon 2} \frac{\varepsilon^2}{k}
\]

Where \( P \) is the shear production, Diff and DDif are the diffusive terms, \( C_{\varepsilon 1} = 1.42 \) and \( C_{\varepsilon 2} \) is a function of the shear rate (not a constant as in the standard \( \kappa - \varepsilon \) model).
**Description of computational implementation.** A rectangular wide channel has been considered with dimensions of 10.0x2.4x1.8 m in x, y and z directions respectively. Periodicity conditions are imposed in x and y directions. In the x-direction (inflow and outflow), periodicity allow us to have a small domain and still have a fully developed flow. In the y-direction, periodicity allows us to consider the channel as wide. At the bottom, a wall boundary has been used and no free surface modeling has been applied for the top boundary. This simplification has been done, since the objective of this study is to discuss about the influence of the probe on the measurement volume (measurement volume located below the instrument) and not on the wake zone. The solid geometry for the probe has been generated in AutoCAD and then converted into 3D solid object (stereolithography).

<table>
<thead>
<tr>
<th>Block</th>
<th>Ext. X (Nx cells)</th>
<th>Ext. Y (Ny cells)</th>
<th>Ext. Z (Nz cells)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>[0.0:10.0] (50)</td>
<td>[-1.2:1.2] (26)</td>
<td>[0.0:1.8] (45)</td>
</tr>
<tr>
<td>2</td>
<td>[4.65:5.75] (40)</td>
<td>[-0.5:0.5] (36)</td>
<td>[0.7:1.55] (44)</td>
</tr>
</tbody>
</table>

Table 1: Mesh characteristics valid for all runs

Figure 1 shows the computational mesh in the XZ plane. A Multiblock (2 blocks) technique has been used (Flow Science, 2003). The main block (standard type) covers the whole channel and the second block (nested type) covers an area around the
instrument location (X=5.0, Y=0.0, Z=1.20 m). Dimensionless plots for mean streamwise velocity (U) and turbulent kinetic energy (TKE) are presented to validate the computations. One can see from figure 2 that the values computed using the present model compares in good agreement with experimental data (Nakagawa et al. 1975) where the variables have been made dimensionless by using the water depth (H), the shear velocity (u*), and the maximum value of the velocity (Um). It is also verified that using Multiblock feature produces the same velocity profile and turbulent quantities (k and ε are the TKE and dissipation of TKE respectively).

![Graph](image.png)

Figure 2: Validation of the model: (a) Velocity profile (b) Turbulent quantities

**Experimental setup**

The wake effect of the ADP Sontek Pulse-Coherent Acoustic Doppler Profiler was studied experimentally through the laboratory measurements in an open channel flume having a length of 420 cm, 40 cm in height and 30 cm in width. A gate located at the downstream extreme of the flume allows modifying the water depth in the flume. Two dimensional velocity field measurements were performed in the centerline of the defined region of the flume with and without the presence of the probe to evaluate the effects of the instrument on the flow field. Particle Image Velocimetry (PIV) was applied to measure the 2-D flow field in both conditions (with and without instrument presence). A 120 mJ double-pulsed Nd:YAG laser was used to illuminate from the transparent bottom a region of the flow field in the center line of the flume. This set up configuration is perfect for this kind of analysis because it allows getting a complete representation of the flow field around the acoustic probe without shadow areas. Image pairs were captured with an interval of 4000 µs between them using a PIVCAM 40-15 Powerview 2000 by 2000 pixel digital camera. The camera had a 50 mm lens and was synchronized with the laser. The size of the
capture field was 33.6 cm by 33.6 cm and a sampling frequency of the 5Hz was used in this analysis. Seeding particles were added to the flow to improve the quality of the captured images. The tested flow condition presented in the analyze region, at progressive 2.30 m were: water depth = 0.32 m, mean velocity = 0.143 m/s and hydraulic radius = 0.103m (Re = 14596).

Despite of the all the effort dedicated to simulate a well developed open-channel flow conditions (logarithmic vertical profile of longitudinal velocities) it could not be obtained. However, the presence of secondary currents in the flow was minimized. The instrument was located at 23 cm from the bottom (at progressive 2.30 m), it means that around 9 cm of the instrument was submerged into the flow.

**Results**

This part summarizes the results obtained by numerical modeling and experiments. These two types of results cannot be compared directly each other so far, since, the domain and flow characteristics are different in each setup; however global patterns are very similar in both cases. Future work in planned to overcome this problem. To measure the influence of the presence of the instrument, two conditions have been identified. The first condition represents the undisturbed flow (without probe) and the second condition represents the disturbed flow (with probe). In order to quantify the influence of the presence of the probe, some variables (given by equation 7) are defined:

\[
U_{cr} = \frac{U_0 - U_I}{U_0} \quad TKE_{cr} = \frac{TKE_0 - TKE_I}{TKE_0}
\]

Where the sub indexes 0 and I represent the undisturbed and disturbed conditions, U, V and W are the cartesian flow velocity components.

**Numerical results.** Three numerical setups have been performed using the same roughness coefficient (Ks) as shown in table 2. Figures 3 and 4 show the velocity magnitude computed as \( \sqrt{U_i^2 + V_i^2 + W_i^2} \) for the disturbed condition in the XZ and YZ planes respectively.

<table>
<thead>
<tr>
<th>Setup</th>
<th>Slope</th>
<th>Ks(m)</th>
<th>Water depth (m)</th>
<th>Mean vel. (m/s)</th>
<th>Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0005</td>
<td>7x10^-4</td>
<td>1.80</td>
<td>1.88</td>
<td>3.33x10^6</td>
</tr>
<tr>
<td>2</td>
<td>0.001</td>
<td>7x10^-3</td>
<td>1.80</td>
<td>2.67</td>
<td>4.71x10^6</td>
</tr>
<tr>
<td>3</td>
<td>0.005</td>
<td>7x10^-3</td>
<td>1.80</td>
<td>5.29</td>
<td>1.05x10^7</td>
</tr>
</tbody>
</table>

Table 2: Numerical setups
Figures 3 and 4 show the velocity magnitude for disturbed conditions in XZ and YZ planes, respectively.

Figures 5 and 6 show the relative streamwise velocity and turbulent kinetic energy as calculated by equation 7. In figure 5, negative contours represent acceleration of the flow due to the presence of the instrument; values between 0 and 1 represent desacceleration of the flow; and values higher than 1 represent that the flow has switch its direction. In figure 6 negative values represent increase in TKE due to the probe wake effect and in the other hand; positive values represent a decrease on TKE.
Figure 6: Relative turbulent kinetic energy, $TKE_{cr}$: (a) RUN1, (b) RUN2, (c) RUN3

**Experimental results.** As in the numerical analysis, the evaluation of the wake effect of the ACP is based in the contrast between the observed flow field present in the flume with and without the presence of the instruments using the parameter defined by equation 7. The groups of images of the flow field for both cases were captured at different times but under the same flow conditions. A very important issue in this analysis is introduced here. Because of measuring in different times, the time series of pair of images must be long enough to reduce the uncertainty in the statistical estimator. Accomplishing that, an analysis is performed which show the evolution of both the mean and the variance of the flow velocity signals for each of the three cartesian components as the number of captured pair of images used to compute it increases. Figure 7(a) shows the behavior of the longitudinal mean velocity value (running mean analysis). The location analyzed in this figure corresponds to a point in the vertical located in the center of the instrument and the point is at 20 cm from the bottom. Figures 7(a) and 7(b) include also (as dashed lines) the intervals of +5 percent and -5 percent of the final statistical value computed using all the figures (3048). Figure 7(b) shows the same analysis for the variance of the flow velocity in the longitudinal direction (used to compute the turbulent kinetic energy of the flow field) at the same location as represented in 7(a). It can be seen that a longer measurement time (higher number of images pairs) is required to obtain a representative value of the variance. The higher the moment of the parameter estimated the longer the required measurement time.

A similar analysis was performed for different points in the flow field. Based in the results, a number of 1000 pair of images was finally chosen as optimum, thus the effect of sampling the flow field at different time is minimzed.
The flow fields were captured for the two flow conditions analyzed here. Then, variables quantifying the wake effects of the instrument in the flow field (equation 7) are plotted in Figures 8(a), (b). Figure 8(a) shows the relative streamwise velocity and figure 8(b) shows the relative turbulent kinetic energy.

Conclusions

The results obtained in the numerical and experimental work indicate that the wake effect of the ADP Sontek Pulse-Coherent Acoustic Doppler Profiler may modify significantly the flow field producing erroneous readings of the mean flow velocities.
and turbulent quantities. These perturbations that the probe produces are not minor, particularly in the separation zone behind the probe.

**Future work**

As it was mentioned before, in the present work there is not a directly comparison of experimental and numerical results, however, a future work is planned. Besides, even though the results (experimentally and numerically) show that the influence of this particular doppler instrument on the flow is less than 10 percent around the probe, there are cases where this influence could be of great importance. For instance, when considering a narrow channel, or when these instruments are located close to walls. Then more investigation is needed to propose some way of correcting the flow field measured. The present methodology could be applied for other acoustic instruments such as the ADV (Acoustic Doppler Velocimeter) and ADCP (Acoustic Doppler Current Profiler) among others.

**References**


