Zentrum für Technomathematik

Fachbereich 3 – Mathematik und Informatik

Simulating the flow deviation of an acoustic doppler velocimeter

Philipp Cordes

Thomas Page

Report 10–06

Berichte aus der Technomathematik

Report 10–06

August 2010

Simulating the flow deviation of an acoustic doppler velocimeter

Philipp Cordes Thomas Page
University of Bremen

2010-06-22

Abstract
The main aim of this study is to quantify possible errors of the measurement of the acoustic doppler velocimeter "Vectrino" due to flow disturbance of the device.
To this end, we use the commercially available CFD software COMSOL Multiphysics ©. We compare the simulation of an undisturbed flow with simulations of flows around the measurement device.
We simulate a one-dimensional, laminar flow close to the seabed. Our investigations show that a difference of up to 30 % of the norm of the velocity occurs in the sampling volume. The difference of the direction of the velocity is up to 27 °.
In the first part of this study we give a short introduction to our problem and note the details of our discretization of the Vectrino. In the second part we present the equations of motion and the boundary conditions of our problem. In the third part we present our results and in the fourth we list some questions for possible further study.

Zusammenfassung
Das Ziel dieser Arbeit ist, mögliche Messfehler des akustischen Dopplergeschwindigkeitsmessgeräts „Vectrino“ zu bestimmen, die durch die Umströmung entstehen.
Dafür haben wir das kommerzielle Softwarepaket COMSOL Multiphysics © verwendet. Wir haben die Simulation eines ungestörten Versuchsaufbaus mit Strömungssimulationen um das Messgerät verglichen.
Die simulierte Strömung ist eindimensional, laminar und nah am Meeresboden. Unsere Untersuchung zeigt, dass im Messbereich eine Abweichung von bis zu 30 % in der Norm der Geschwindigkeit und eine Winkelabweichung von bis zu 27 ° auftritt.
Im ersten Teil geben wir neben einer kurzen Einführung in unsere Problemstellung die Diskretisierungsdetails an. Danach widmen wir uns den verwendeten Strömungsgleichungen und Randbedingungen. Im dritten Teil stellen wir die Ergebnisse vor und geben schließlich einen Ausblick auf weitere Fragestellungen.

Keywords
Acoustic doppler velocimeter, finite element method, laminar flow, one-dimensional flow

Acknowledgements
We want to thank Prof. Michael Böhm of the Center for Industrial Mathematics at the University of Bremen and Prof. Arzhang Khalili of the Max Planck Intitute for Marine Microbiology in Bremen for their support during the study.

1 Introduction

Numerical simulations contribute greatly in the description and anaylsis of problems in nature and technology. Nevertheless there is the need for experiments to validate these numerical results and methods.
One possibility to validate results of the flow velocity is given through acoustic doppler velocimeters. ADVs are commonly used in the marine community to measure flow near sea beds. These

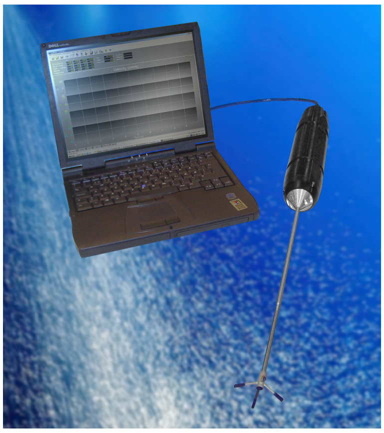


Figure 1: The acoustic doppler velocimeter: Vectrino

invasive measurement devices employ the doppler effect to measure the velocity. During the measurement part of the device is placed into the flow close to the sampling volume. The device sends acoustic signals, which are reflected by scattered material in the water. Due to the doppler effect these signals shift according to the speed of the reflecting particle. The device measures this shift and can determine the speed of the particle, which is assumed to be the same as the speed of the fluid.
The sampling volume is below the head of the device (figure 2). Our main interest is the bias in the sampling volume.
Experimental studies have shown that the error introduced by accoustic doppler velocimeters is not neglectable and also dependend on the orientation of the measurement arms to the flow [3]. Our study is meant to quantify these errors with a CFD approach to this problem.
For our investigations we chose to simulate a one-dimensional laminar flow close to the sea bed.

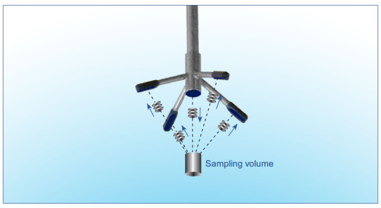


Figure 2: The Vectrino head and sampling volume

1.1 Geometry

We limit our study to the bottom part of the measurement device since we are mostly interested in the flow in the sampling volume. We approximated the Vectrino with cylinders and frustums of cones (see figure 3 for details) and accepted small errors for this discretization. The four arms are pitched down by $30\,^{\circ}$.

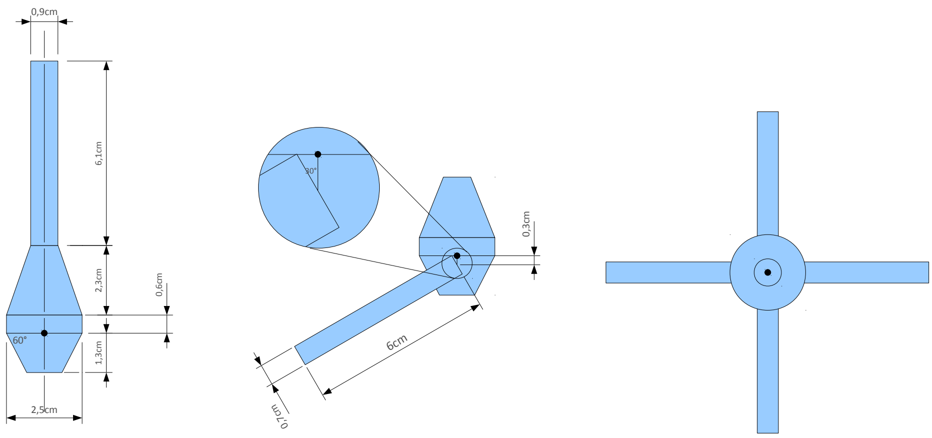


Figure 3: The discretized Vectrino

2 Mathematical formulation

In this section we present the equations of motion and the boundary conditions for the setting described above. We viewed this problem in the context of continuum mechanics.

2.1 Equations of motion

For our setting we make some assumptions:

1. The temperature is constant.
2. The fluid is homogeneous and has a constant density.
   We neglect the scattered material which is needed for the doppler effect.
3. The flow is laminar and stationary.

With these assumptions the following system of equations describes the balances of momentum and mass. These have to be fulfilled inside our domain and are the partial differential equations we have to solve. As the temperature is assumed constant we do not consider the energy equation.

- **Continuity equation**

$$\nabla \cdot u = 0$$

- **Navier-Stokes equations**

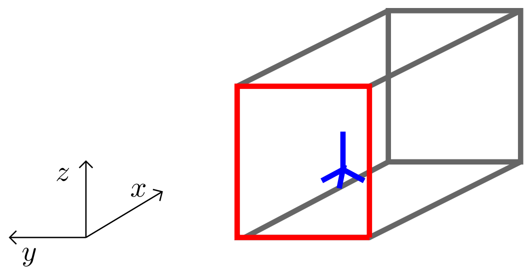
$$(u \cdot \nabla)u = -\frac{1}{\varrho}\nabla p + \nu \Delta u$$

$u \in C^2\left(\mathbb{R}^3, \mathbb{R}^3\right)$ is the velocity field, p $\in C^1\left(\mathbb{R}^3, \mathbb{R}\right)$ is the pressure, $0 < \varrho \in \mathbb{R}$ is the density and $\nu \in \mathbb{R}$ is the kinematic viscosity.

2.2 Domain and Boundary Conditions

As our domain for the simulation we chose a simple box (see sketch below). The bottom of the box is supposed to be the sea bed. The flow at the inlet, marked in red, is a one-dimensional flow in x-direction. The Vectrino, marked in blue, is modelled as a solid object which can not move. The Vectrino and the bottom are both modelled as solid walls. The head of the Vectrino is 10 cm away from the sea bed. The other four boundaries, the top, the two sides and the back, are modelled as open boundaries. They are supposed to have no influence on the flow.

We consider every real boundary to be so far away from our setting that it has no influence on the flow inside our domain.



The specific measures are

- depth: 75 cm (x-direction)
- width: 40 cm (y)
- height: 63.7 cm (z)

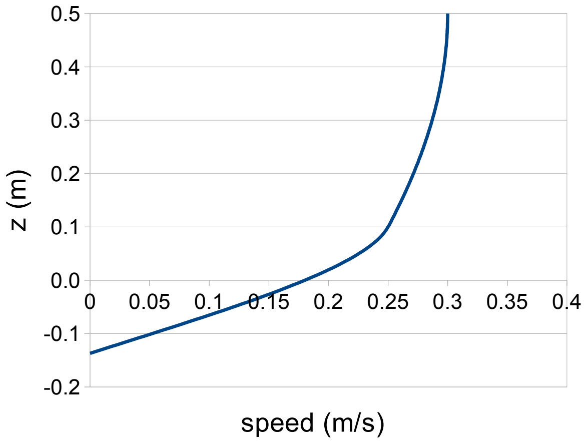


Figure 4: The inlet profile, the seabed is at $z = -0.137$.

The boundary conditions

1. The bottom wall and the Vectrino are modelled as a solid walls. We therefore chose the no-slip boundary condition:
$$u(x, y, z) = 0$$

2. At the inlet (marked in red) we set an inlet profile:
$$u(x, y, z) = u_{\text{inlet}}(z)$$
In figure 4 the inlet profile is shown. The y-axis represents the height of our simulation box and the x-axis the velocity in x-direction. The inlet velocity is constant in y-direction. The velocity at the bottom is zero and the velocity at the top is 0.3 $\frac{m}{s}$ in x-direction. To compute this profil we simulated a flow for a long empty channel with the same cross sectional area and boundary conditions as our problem. We guessed an almost quadratic inlet for this setting and then took the outlet of this simulation as the inlet in figure 4.

3. For the remaining boundaries (top, outlet and the two sides) we chose the open-no vicous stress boundary. This reflects our assumption, that these boundaries have no influence on the flow and no viscous stress is brought into our domain:
$$\underbrace{\frac{1}{2}\eta\left(\nabla u + (\nabla u)^T\right)}_{\text{viscous stress tensor}} \cdot n = 0 \qquad \eta \in \mathbb{R}$$

4. We also set the reference pressure at the outlet:

$$p(x, y, z) = 0$$

2.3 Reynolds number

In our setting the Reynolds number is

$$Re = \frac{\rho \cdot U \cdot L}{\nu} = 5800.$$

where $U = 0.232 \frac{m}{s}$ is the average velocity of our inlet, L = 10 cm is the distance of the head to the floor and $\rho = 250 \frac{kg}{m^3}$ and $\nu = 0,001 Pa\ s$ are material properties of our simulation fluid. For water this is equivalent to a characteristic speed U = 5.8 $\frac{cm}{s}$.
To solve this problem we used the commercially available software COMSOL Multiphysics ©. A typical mesh of our domain is shown in figure 5. We chose a finer mesh at the areas of

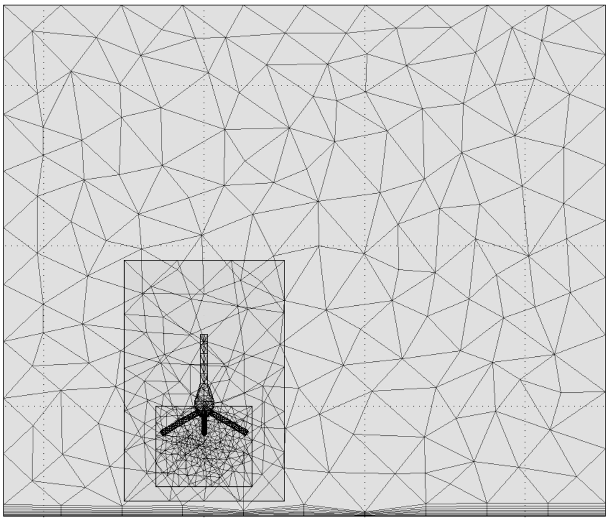


Figure 5: A typical mesh with the three differently meshed subdomains

our interest and where we expected high gradients. Therefore we divided the domain into three subdomains. We chose a very fine mesh near the sampling volume, a slightly more coarse one around the rest of the velocimeter and an even coarser one for the rest of the domain. Also we chose a fine boundary layer mesh at the floor and at the Velocimeter. We did this because of the high gradients coming from the no slip boundary condition.

3 Results

Our results show that the Vectrino has a significant influence on the flow. The orientation also has a major impact on the deviation caused. One gets an idea of how the numerical solution looks like in figure 6. The inlet is on the left side of each picture, the outlet on the right. The color corresponds to the absolute value of the flow speed u. Dark blue means a speed of 0 $\frac{m}{s}$ and dark red is the maximum speed of 0.3 $\frac{m}{s}$. The sampling volume is marked with a small white outline. The disturbance is much greater behind the device and close to its borders. We concentrated our work on the sampling volume below the Vectrino. If you look closely you can see a yellow stripe below the device indicating an increased velocity. While it is rather small in the left picture, you can clearly observe it in the right one. The pictures suggests that the device has an influence on the flow inside the sampling volume and that both depicted influences are different in value. We were able to confirm this.

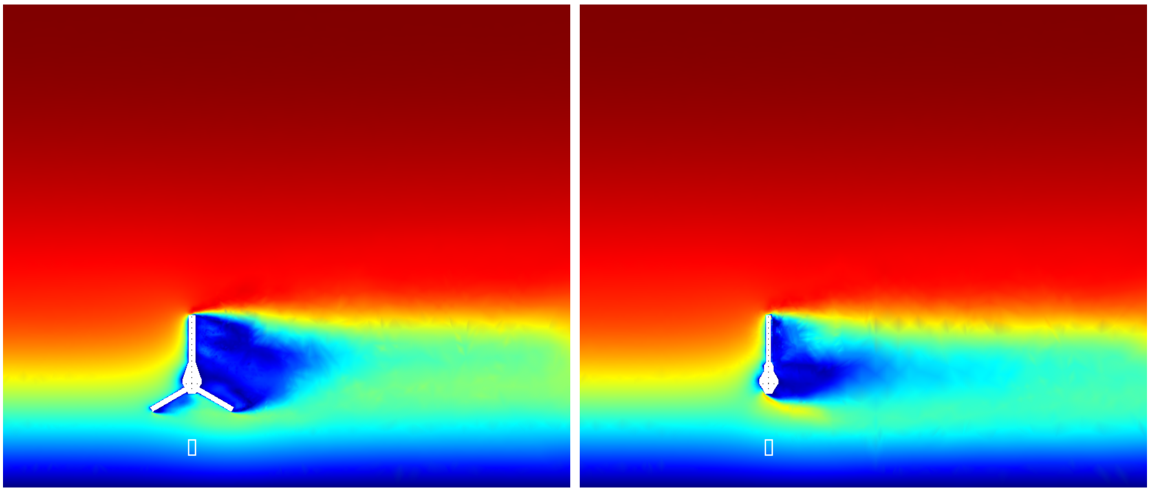


Figure 6: Side view of the central slice of the velocity field for cases two and three

3.1 Parameter Variation

We studied the solutions under two aspects. The first aspect was how mesh dependend the solutions are and how they change for finer meshs. The second one was how the simulations differ due to the orientation of the Vetrino to the flow. To do this we simulated three different

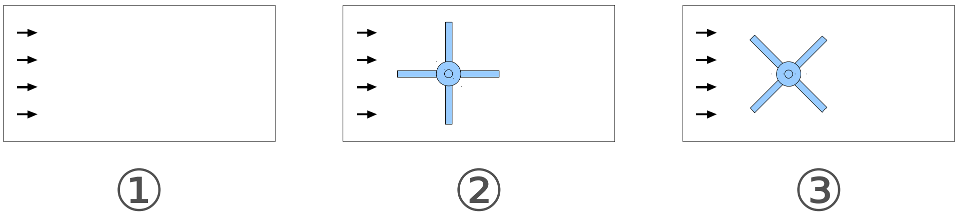


Figure 7: The three cases we simulated, viewed from above

cases (see figure 7):

1. The empty domain, for reference purposes
2. The channel with the Vectrino
3. The channel with the Vectrino rotated by 45 °.

We meshed the empty channel with cuboids. The inlet profile was basically continued through the entire domain. We considered this to be the true undisturbed flow and used it as a reference.
The other cases were meshed using tetrahedral meshes. They were created using three subdomains, as described above (see page 5).

3.1.1 Mesh dependency

For Situations two and three we created three meshes different in coarseness and investigated on their influence. The finer meshes consist of more elements both globally and especially in subdomain three (that contains the sampling volume). You get an impression of the coarseness in table 1. The total element numbers range from about 15 000 to more than 100 000. We then compared the velocity components for each situation in the sampling volume.

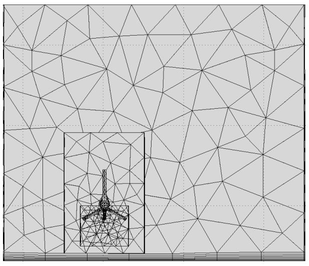
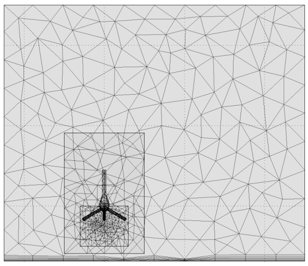
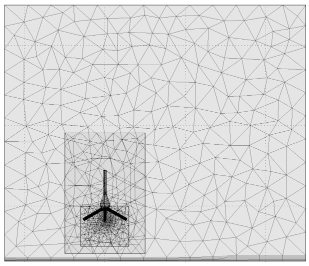
			
	coarse	medium	fine
mesh elements in domain	15 438	51 359	101 683
mesh elements in subdomain 3	6 375	29 768	67 219

Table 1: Meshes for situation two

The sampling volume is a cylinder $5\,cm$ below the Vectrino and its height can be adjusted between $3\,mm$ and $15\,mm$. It has a diameter of $6\,mm$. We discretized this volume by using a cubic mm grid and then averaged the values over their z coordinate. This negate symmetric effects caused by the shape of the Vectrino. This yielded z coordinates between -0.063 and -0.078, they are used for the horizontal axis of most diagrams to come.

The velocity components of situation three are depicted in figure 8. The red lines stand for the results of the coarse mesh, the yellow ones for the medium mesh and the green lines for the fine mesh.

The results for the velocity components u_x and u_z for the middle and fine meshes (yellow and green line) are very similar. The values of u_y are rather uncomforting, but are much smaller and almost zero. They nearly don't contribute to the absolute deviation as they are two or three magnitudes smaller than the other components. This leads us to the assumption that further refinements of the mesh will not lead to significantly different results. The same applies for the solutions of situation two. Therefore we consider our solutions sufficient for further analysis.
For our second aspect of anaylsis, the influence of the orientation, it was nessecary to compare two velocity fields.

How we compared two velocity fields. As described above (paragraph "The sampling volume" on page 7), the results are averaged over z. For each z we have the three speed components u_x, u_y and u_z. Although they are important, they are rather unhandy concerning a quick overview of the amount of deviation. This is due to them being an order of magnitude apart from each other ($|u_x| \approx 10\,|u_z| \approx 1000\,|u_y|$).

A simple way of comparing these velocity fields is to compare for each point both the absolute value and the angle between the vectors.

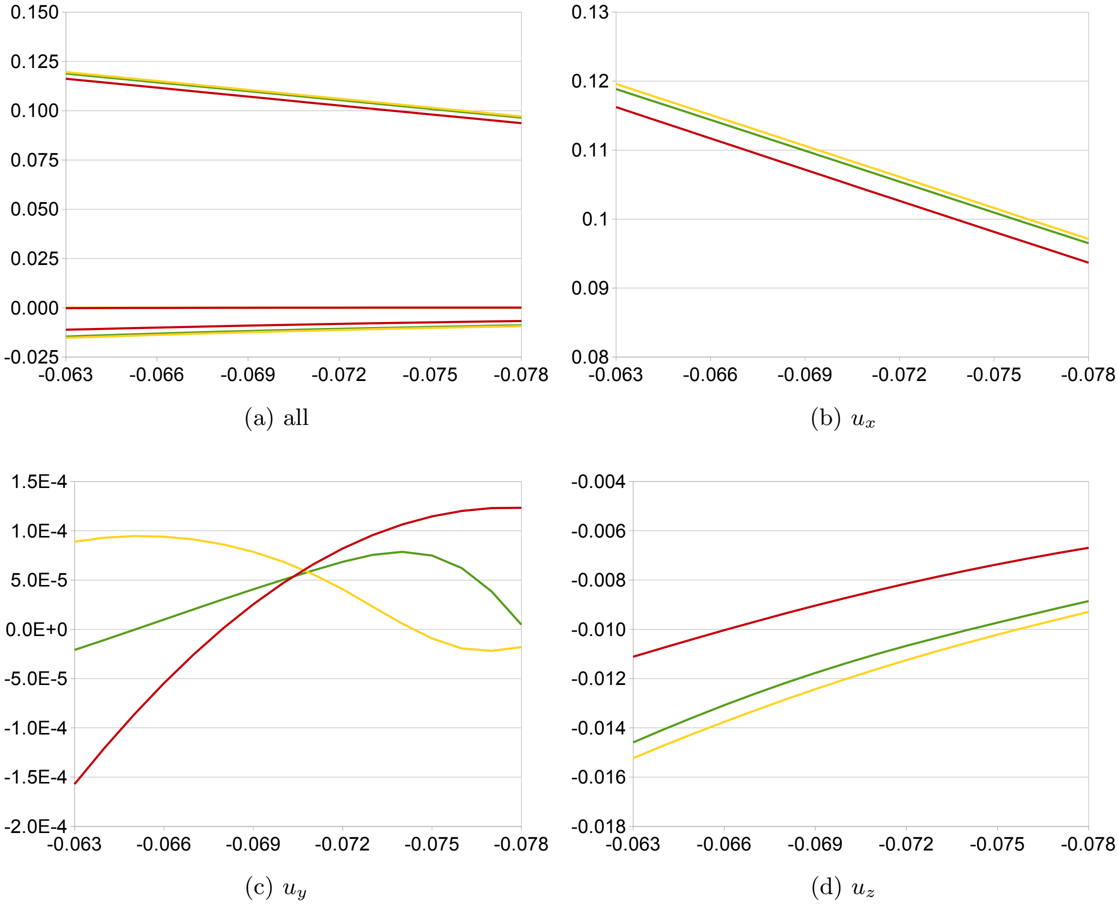
3.1.2 Influence of orientation

For situations two and three we computed

$$d = \frac{|u| - |\overline{u}|}{|\overline{u}|} \qquad\qquad \varphi = \arccos \frac{u * \overline{u}}{|u| \cdot |\overline{u}|}$$

to compare the velocity field with the reference values $\overline{u}$ from situation one. Here $u * \overline{u}$ is the euclidean scalar product. d is the relative difference of the absolute value of the flow speed and φ is the angle deviation. As supposed in the introduction to this section (see page 5) the influences of the Vectrino in situations two and three are different.

In figures 9 and 10 the blue line stands for situation two and the red line for situation three. The graph for situation three is sensible: The deviation decreases when farther away from the



(a) all (b) u_x (c) u_y (d) u_z

Figure 8: Velocity components of situation three for all meshes in $\frac{m}{s}$, together and split up

device. But its value is very large: The flow is 27 % faster at the top of the sampling volume. At its bottom it's still 17 % faster. The angle shows the same behavior: Its devation ranges from 27 °at the top to 15 °at the bottom. The direction of the angle can be derived using the raw data. The flow isn't parallel to the floor any more, it points down by the respective angle.

In the second case the flow is also perturbed, but not that much: The deviations are rather constant at about 15 % and 6 °. This can be explained by the arm that is directly in the way of the flow which slows it therefore down, negating the influence of the Vectrino.

In conclusionAs a summery: The Vectrino influences the flow in our simulations a lot. This may not be a universally valid approach as we employed many simplifications. A big problem though is the difference between the two situations. It is difficult to correct such different biases even for such a simple situation. At least it is important to keep the same orientation to avoid adding additional noise to the measurement and to keep results comparable.

4 Future

Our study shows that the results given through the Vectrino have to be considered with care. Influences up to 30 % in the norm of the velocity are obviously critical. To give a full evaluation there are several further aspects that need to be considered:

1. How big is the influence if the flow is turbulent? The Vectrino is often used to measure the velocity of turbulent flow. We only studied the laminar case which might have very different results than the turbulent case.
2. Compare data the Vectrino delivers with the simulations. It is necessary to investigate which errors are corrected by the Vectrino companion software and how they are corrected.

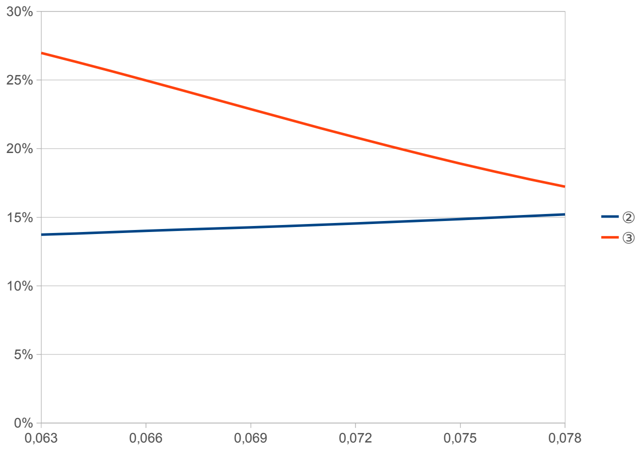


Figure 9: absolute value deviation

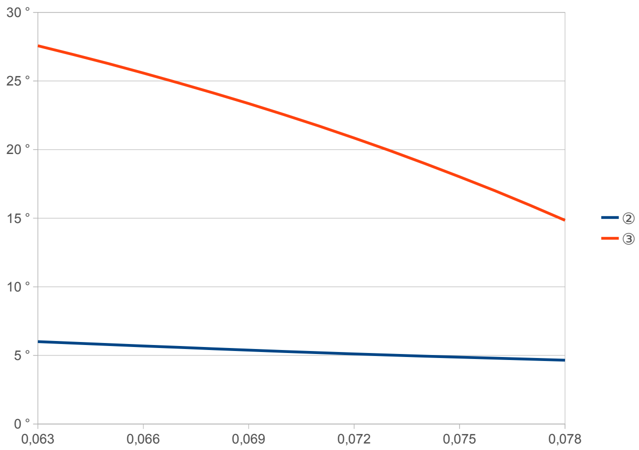


Figure 10: angle deviation

3. Study more orientations of the device. We only studied two different settings. It is important to know which are the critical orientations. On the one hand it is of interest how the different orientations influence the flow and on the other hand how the software corrects these errors for each orientation.

4. Study more complex settings. How does the Vectrino influence the flow if one or more of our assumptions (the flow is one-dimensional, stationary, the fluid is homogen, the temperature is constant and the seabed is modelled as a plain wall) are not made?

References

[1] Nortek AS: **Vectrino – A new generation 3D water velocity sensor!**
Manufacturer's brochure about the Vectrino
`http://www.nortek-as.com/lib/brochures/Vectrino%2006%20b.pdf/view` (accessed on 2009-10-23)

[2] Nortek AS: **Vectrino User Guide** (October 2004, Rev. C)

[3] Peter J. Rusello, Atle Lohrmann, Eric Siegel, Tim Maddux: **Improvements in acoustic doppler velocimetry**

The 7th Int. Conf. on Hydroscience and Engineering (ICHE-2006), 10^{th} – 13^{th} September 2006, Philadelphia, USA.

[4] Herbert Oertel, Eckart Laurien: **Numerische Strömungsmechanik: Grundgleichungen und Modelle - Lösungsmethoden - Qualität und Genauigkeit**, 3. Auflage
Vieweg+Teubner Verlag, Wiesbaden 2009, ISBN: 978-3-8348-9964-4

[5] Joseph H. Spurk, Nuri Aksel: **Strömungslehre – Einführung in die Theorie der Strömungen**, 7. Auflage
Springer Verlag, Berlin, Heidelberg 2007, ISBN: 978-3-540-38441-0