A Study on the Flow of Viscous Fluids in a Square Duct

Emilia Rieschel
emilia.rieschel@gmail.com

under the direction of
Prof. Luca Brandt
Department of Fluid Mechanics
Royal Institute of Technology

Research Academy for Young Scientists
July 13, 2016
Abstract

Fluid flow in circular and noncircular pipes is commonly encountered in practice. The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines. An understanding of the flow in pipes and ducts would therefore help us to make the flow as efficient as possible. Of the reason that the studies in square ducts are few, the aim of this study is to study the flow of viscous fluids in a square duct. To do this a technique called Particle Image Velocimetry (PIV) has been used, working non-intrusively and measuring the velocity indirectly. The results shows that the flow is asymmetric in laminar regime but, becomes symmetric in turbulent regime due to intense mixing of the fluid. This indicates the amount of care needed to be taken to build an ideal set-up. As the set-up was not ideal, no conclusions could be made from the results. In the future however, the behavior of the particles is to be observed in turbulence, so the set-up may serve its purpose.


1 Introduction

Human beings have always been interested in observing movement, as it in many cases implies danger, and consequently, an understanding of movement is of most importance for their survival. The movement of fluids, or the so called flow, became especially interesting when the first cities were built, as it was necessary to develop water pipeline systems. Thus the study of flow, fluid mechanics, was born in the Antiquity. However, until the last centuries experimental tools were limited, and it was only possible to describe qualitative properties. One of the most famous observers of fluids throughout history was Leonardo da Vinci. He was able to study the turbulence in fluids through mere observation. One of da Vinci’s drawings can be seen in Figure 1. The technical progress of the last decade, especially of lasers, video, computer technologies and optics has resulted in experimental methods that can visualize the qualitative flow to such an extent that the same methods also can be used for quantitative measurements. This has made it possible to study fluid mechanics in more detail and thus understanding how it works. [1, 2]

Figure 1: Leonardo da Vinci’s Drawing.  

\footnote{Sketch of turbulence by Leonardo da Vinci ca. 1510, http://mbalajew.ae.illinois.edu/research.html}
1.1 Aim of the Study

Fluid flow in circular and noncircular ducts is commonly encountered in practice, and an understanding of the behaviour of flow in ducts would therefore make it possible to make the flow in those ducts as efficient as possible. However, flow, both inside and outside ducts, are rarely completely pure from other particles than the fluid’s own. As a result, if the flow of particles in a fluid can be understood, a better understanding of the behavior of fluids in ducts in general will be attained. To be able to study the flow of particles, a reference case is at first needed, where the flow is studied in a fluid only with particles that have a negligible effect on the flow.

Since research on flow in square ducts is few, this study will be performed in one. Thus, the aim of this study is to investigate the flow in a viscous fluid in a square duct, without particles disturbing the flow.

1.2 Theory

To be able to understand this study, a general description of internal flow is needed. Initially, the foundations of flow will be explained, followed by a discussion of the dimensionless Reynolds number and its physical significance. Then it will be explained how the flow differ through the duct theoretically and finally how the velocity profile can be calculated using simplified Navier Stoke’s equations.

1.2.1 Foundations of Flows

Flows can be divided into laminar and turbulent flow, where smooth streamlines and highly ordered motion characterizes laminar flow, while velocity fluctuation and highly disordered motion characterizes turbulent flow. Generally, laminar flow occur at low velocities and turbulent at high velocities. The transition from laminar to turbulent flow occurs when a small disturbance in the flow no longer can be neglected as the flow inertia becomes too significant relative to the fluid viscosity. [4]
The no-slip condition states that all fluid particles in contact with a surface comes to a complete stop and assumes zero velocity relative to the surface. The reason for this is that when a fluid moves along a solid boundary, a shear stress will act on the fluid. The definition of shear stress $\tau$, is tangential force $F$ per unit area $A$,

$$\tau = \frac{F}{A}. \quad (1)$$

When the layers of fluid in contact with the surface of the duct slow down, they cause the fluid particles in the adjacent layers to slow down gradually as a result of friction. To compensate for this velocity reduction, the velocity of the fluid at the midsection of the pipe has to increase to keep the mass flow rate through the pipe constant. Assuming the fluid is incompressible, the mass flow is constant because otherwise it would mean that less mass comes out from the duct than in, resulting in that mass disappear. The several layers of the fluid will consequently all have different velocities, and a velocity gradient develops along the duct. The velocity gradient indicates the velocity difference between adjacent layers. It is written as $\frac{\Delta u}{\Delta y}$, where $\Delta u$ is the velocity difference between the layers and $\Delta y$ is the distance between the layers. [2]

### 1.2.2 Reynolds Number

During the end of the 19th century Osborne Reynolds studied the transition from laminar to turbulent flow, which resulted in Reynolds number. Reynolds number is the ratio of the inertial forces and the viscous forces:

$$Re = \frac{\text{Inertial Forces}}{\text{Viscous Forces}} = \frac{\rho VL}{\mu}, \quad (2)$$

where $\rho$ is density of the fluid, $V$ the bulk velocity of the duct’s cross section, $L$ the diameter of the duct and $\mu$ the dynamic viscosity. The bulk velocity, also called the
average velocity, depends on the flow rate $Q$ and the cross section area $A$,

$$V = \frac{Q}{A}.$$  

The Reynolds number can be used to determine if the flow is laminar, turbulent or in transition between these regions. Different geometries have different critical Reynolds number, meaning the flow is in transition at different Reynolds numbers in different geometries. In a circular duct the following applies; when $Re \leq 2000$ the flow is laminar, and when $Re \geq 4000$ the flow is turbulent. [5, 6, 7, 8]

To calculate the Reynolds number, the viscosity of the fluid is needed. The viscosity is a measure of a fluid’s resistance to deformation, occurring due to internal frictional forces between different layers of the fluid as they are forced to move relative to each other. To determine the viscosity, a constitutive relationship has to be used. In that relationship the shear stress and velocity gradient are included. The shear stress causes the velocity gradient, and if the shear stress increases, the velocity gradient increases. Consequently, for all Newtonian fluids in laminar flow the shear stress, which can be calculated as in equation 1, is proportional to the velocity gradient;

$$\tau \propto \frac{\partial u}{\partial y}.$$  

This proportionality can be used to calculate the Reynolds number. Earlier experiments have shown that the constant of proportionality is the viscosity $\mu$ [2, 5, 6], resulting in:

$$\tau = \mu \frac{\partial u}{\partial y}.$$  

1.2.3 Development of Flow

When a fluid enters a duct, the particles in contact with the boundary will slow down according to the no-slip condition. The region of the flow in which the effects of the viscous shearing forces are felt is called the velocity boundary layer. The hypothetical
boundary surface divides the flow in two regions; the boundary layer region, in which the viscous effects and the velocity changes are significant, and the irrotational flow region, in which the frictional effects are negligible and the velocity remains essentially constant in the radial direction (if the geometry under consideration is a circular duct). [2]

As can be seen in Figure 2, the thickness of this boundary layer increases in the flow direction until the boundary layer reaches the duct center and fills the entire duct. The region from the duct inlet to the point at which the boundary layer merges at the centerline is called the hydrodynamic entrance region, and the length of this region is called the entrance length. Flow in the entrance region is called developing flow since this is the region where the velocity profile develops. The region beyond the entrance region in which the velocity profile is fully developed and remains unchanged is called the fully developed region. [2]

![Figure 2: Entrance Length.](image)

---

The entrance length for a circular duct, can be estimated using the equation

\[ \frac{L_e}{D} = 0.057 R_e, \]  

where \( L_e \) is the entrance length, \( R_e \) is the Reynolds number and \( D \) is the diameter of the duct. For a square duct, there is no equation for the entrance length. However, equation 3 can be used, and by making experiments, be compared to experimental results. For non-circular geometries, the hydraulic diameter \( D_H \) can be used as \( D \) in equation 3, which is calculated by;

\[ D_H = \frac{4A}{P}. \]

Where \( A \) is the cross-sectional area of the duct and \( P \) is the circumference of the wet part of the duct. [4]

1.2.4 Navier-Stokes Equations

The motion of viscous fluids can not only be described by experiments, but also theoretically by equations. The most important equations visualizing flow are the Navier-Stokes Equations, which, however, can be solved analytically only for a few simplified cases. By simplifying the Navier-Stokes equations, assuming that the fluid has constant physical properties, that natural convection is negligible and that the pressure gradient along the same direction as the flow is constant, it can be used to describe the flow in a duct. The simplified Navier-Stokes equation in dimensionless form is:

\[ \frac{\partial^2 V}{\partial x^2} + \frac{\partial^2 V}{\partial y^2} + 1 = 0, \]

where \( V \) is the dimensionless velocity that depends on \( x \) and \( y \). [9]

M. Piga and G.L. Morini from University of Bologna [9] have, using the mentioned assumptions and the simplified Navier-Stokes Equation, calculated an expression that can be used to provide the velocity distribution in a flow in a duct like in Figure 3, also
called the velocity profile:

\[
V(x, y) = \frac{16}{\pi} \sum_{n \text{ odd}}^{\infty} \sum_{m \text{ odd}}^{\infty} \frac{\sin(n\pi x) \sin(m\pi y)}{nm(n^2 + m^2)}.
\]

To be able to apply this equation on reality, Figure 3 and the following equations are used; \( x = \xi/a (0 \leq x \leq 1) \) and \( y = \eta/a (0 \leq x \leq 1) \), with \( \xi, \eta \) and \( a \) defined as in Figure 3.

![Figure 3: A Dimensionless Square Duct.](image)

### 1.3 Particle Image Velocimetry

Particle Image Velocimetry, or PIV, is a technique that measures flows. The idea of PIV is that tracer particles are added to the flow. The requirements of these tracer particles are that they have approximately the same density as the fluid, and are very small. By studying the movement of the particles, the instantaneous velocity of the fluid can be measured indirectly. This is done by using a laser to illuminate the particles in a plane of the flow twice within a short time interval. The light scattered by the particles is then captured with a high speed camera on two consecutive frames. Using the images, cross-correlation algorithms of PIV systems determine the velocities at thousands of area elements called interrogation regions throughout the entire plane and display the velocity.
field in any desired form. The difference between PIV and other velocity measurement techniques is that PIV, using optical techniques, works non-intrusively and measures the velocity indirectly. It is one of the techniques that allows recording of images of large parts of flow fields, instead of only analyzing the movement of one particle in the flow. How a setup with PIV is constructed can be seen in Figure 4. [1, 2]

2 Method

As mentioned, the technique used to study the flow is PIV. Initially in this section, the setup will be explained, followed by a description of the data collection and processing. At last the methods used for specific experiments are thoroughly explained.

2.1 Setup

The setup used in this study can be seen in Figure 5, and the foundation is a 5000 mm x 50 mm x 50 mm square duct, completely filled with tap water. In Figure 5a, most of the duct is shown together with the tank and the velocity meter. The fluid is transported
from the tank to the square duct through a circular duct. The conversion from the circular to the square duct occurs through a 20 cm long tube which is gradually converted into a square. To get the fluid moving, a pump is connected to the end of the duct, pumping the fluid through a filter and then back to the duct. In Figure 5b, the movable platform with the laser and camera is shown. Underneath the duct is rails for the platform, meaning that the flow can be measured in all positions of the duct. The model of the laser used is MGL-W-532A, having a wavelength of 532 mm and is a continuous wave (CW) laser, meaning that light is continuously emitted from it. The camera used is from Teledyne Dalsa and of the model Genie HM1024. The resolution is 1024 x 768 pixels and the maximum frame rate is 117 frames per second.

![Image](image1.png)

(a) An overview of the setup used in this study, where the tank can be seen in the background of the image, and the velocity meter in the upper right corner.

![Image](image2.png)

(b) The movable platform with the laser and camera.

Figure 5: The setup used in this study.

### 2.2 Data Collection

Initially tracer particles, Polyanid Seeding Particles (PSP) with a size of 20 µm, were blended in the water of the tank. Then by using the equation for the Reynolds number (equation 2), as well by visual observation, the flow rate was calculated to be less than 6 liters/min for it to be laminar, and higher than 15 liters/min for it to be completely
turbulent. Thus when experiments were done on laminar flow, a flow rate of 3.5 liters/min was used, and when done on turbulent flow, a flow rate of 16 liters/min was used. The platform with the laser and camera was moved so that the center of the illuminated plane is where the wanted measurement would be performed. Using the software PIV Labview, the number of images, the time interval between the images and how many sequences of images to be taken, were selected. The time interval has two requirements; the displacement of the tracer particles should be less than $1/4$ the size of the interrogation area simultaneously as it has to be long enough to be able to determine the displacement between the images of the tracer particles with sufficient resolution. The interrogation area was 32 pixels, therefore the displacement should be less than 8 pixels between two images. The time interval used for laminar flow was 2 seconds.

2.3 Data Processing

Data was processed in MATLAB, see Appendix. Initially, the area of measurement was specified by marking were it starts and ends with intervals, seen in Figure 6. For each image sequence two intervals were made, resulting in a mean value of the measurement area and also a calibration of the conversion from pixels to mm. Due to bubbles, reflection at edges and scratches on the plexiglass, the raw image has some disturbances. To remove them, the background image was calculated by in each point taking the minimum of the intensity in all images and subtract it from the same point in all the images. The displacement length of the particles was used together with the known time differences to calculate the velocity of the particles. Each sequence could in this way be converted to a velocity profile.
2.4 Navier-Stokes Calculations

The velocity profile was also calculated using Navier-Stokes Equations. The result was then visualized in a three-dimensional diagram.

2.5 Analyzing the Entrance Length

To determine how the flow develops in the entrance region, the entrance length was at first calculated as if the duct was circular, by using the hydraulic diameter in equation 3. To analyze the entrance length seven places distributed along the duct were chosen where measurements would be made, seen in Figure 7. Two measurements were performed at each position; the first measurement was 50 sequences of two images and 2 seconds between sequences and the second measurement, 30 sequences of two images and 1 second between sequences.

Figure 7: Measurements of the entrance length, where the green boxes symbolize the studied planes.
2.6 Three-dimensional Velocity Profile

To obtain a three-dimensional velocity profile of laminar flow by doing experiments, PIV was used. Instead of only performing the experiment on a plane just in the center of the tube, as with the other experiments, it was made at 10 locations in total in the tube. The measurements were at the same distance of length, but with 5 mm difference widthwise, that is if the duct is 50 mm wide, measurements were done at the following planes: -20 mm, -15 mm, -10 mm, -5 mm, 0 mm, +5 mm, +10 mm, +15 mm, +20 mm and no-slip boundary condition was used at -25 mm and +25 mm. The distance from the duct’s entrance was 374 cm, since it could be secured that the velocity profile was fully developed at this position. The velocity profiles could then be assembled into a three dimensional velocity profile.

3 Results

When calculating the Entrance Length for a circular duct but using the hydraulic diameter for the square duct, the value 350 cm was received for a flow rate of 3.5 liters/min. It was assumed that for a square duct, this value would not change a lot. In Figure 8, fully developed velocity profiles for both laminar and turbulent flow can be seen. In the

![Figure 8: Velocity Profiles 374 cm from the Entrance.](image-url)
velocity profile of laminar flow, seen in Figure 8a, the velocity peak is not in the center of the duct. Instead it is approximately 5 mm above. However, in Figure 8b, the velocity peak is in the center of the duct. Additionally, it can be seen that the velocity in turbulent flow is higher and that the velocity profile never reaches a velocity of 0 mm/s. Figure 9 shows two three-dimensional diagrams, visualizing the velocity profile at the same place of the duct. As is evident, the velocity peak is not at the center of the duct, but instead, approximately 5 mm to the left of the center. Only the first and last of the measurements from the analysis of the entrance length can be seen in Figure 10, since the rest of the
velocity profiles look the same.

4 Discussion

As can be seen in Figure 8a, 9b and 10, the velocity peak is not in the center of the duct. Meaning, the results does not coincide with theory, which can most clearly be seen when the three-dimensional plot (Figure 9b) is compared with the plot received from the Navier-Stokes equation (Figure 9a). The asymmetry in the mentioned velocity profiles indicates that there is a source of error in the duct that has been used. However, as can be seen in Figure 8b, the peak of the velocity profile for the turbulent flow is in the center of the duct. The reason for this is probably that during turbulence, the fluid is mixing in all directions resulting in the velocity being almost homogeneous in the whole duct. Regarding the entrance length, the velocity profiles look similar both in the beginning of the duct and in the end, meaning that the results indicate that the velocity profile is already fully developed at 78 cm from the entrance. Since the results shows the flow is already fully developed in the first measurement, no conclusions can be made of what the entrance length is. Additionally, the entrance length for a circular duct with the same conditions was calculated to 350 cm. Logical deduction leads to the conclusion that the difference between those values are significant big, meaning the asymmetry probably affected the entrance length too, making it shorter.

4.1 Sources of Error

As the duct was supposed to behave as a perfectly symmetrical square duct, the velocity profile should also be symmetrical. The reason why these results are obtained have been further investigated. One option was that the pump could be the source of error. To investigate whether this was the cause, the pump was shut off and the flow was disconnected from the pump by opening valves at the ends of the duct. The flow still continued due to the pressure in the tank and gravity. However, no more water was added to the tank, re-
sulting in a reduction of the amount water in the duct when the tank was emptied. Thus, the measurements, when doing this experiment, can only be made in a short interval. An asymmetry was still present in the results, which meant the pump was not the source of error.

Another source could be that the duct was not entirely horizontal. This was investigated by increasing the height of the entrance of the duct with 10 mm, and then examining the flow again. However, the asymmetry was as significant as before. This means that the tube not being perfectly horizontal is not the source of error to the asymmetry of the velocity profile.

A further option is that a swirl can develop due to the duct bend between the tube and the tank. The asymmetry was investigated by filling the entrance of the duct with 0.6 cm in diameter, 12 cm long straws. This resulted in the water flowing through the straws, hence forced into a laminar flow. However, the asymmetry still existed in the velocity profiles, which means that the bend in the duct was not the source of asymmetry.

Another alternative to the asymmetry, which has not been investigated, is using a sponge in the inlet of the duct. This would reduce disturbance, meaning, that if the cause of the asymmetry is that there is some disturbance in the inlet, a thicker sponge could solve that problem. However, the thicker the sponge, the more it reduces the pressure in the duct, resulting in an increase of energy demand.

The problem causing the asymmetry could also be the conversion from a circular duct to a square duct. Although there is a linear transition, 20 cm is a short distance, which means that the transformation may be too fast. This would result in disturbances in the flow, which could explain the asymmetry in the velocity profiles.

To be able to detect when and where the asymmetry starts, the flow can be observed in the inlet of the duct. This can be done by replacing the current parts between the tank and the duct with glass parts so that PIV also can be performed at the inlet of the duct.
4.2 Conclusion

In the square duct used in this study there is a source of error, resulting in an asymmetry of the velocity profiles when examining laminar flow. As the source of error could not be found, the impact of it could not be specified. In conclusion, the results are not reliable.

4.3 Future Perspective

In future studies, this study may be repeated but with a duct without asymmetry, which would result in a reference case of the behavior of the flow in a square duct. Further studies beyond that would be to study the flow of larger particles, which was the original purpose of the experimental facility that was used.

Additionally, further studies can be done, for example examining how the flow is affected by temperature changes. Logical deduction leads to the conclusion that at a higher temperature, more disturbance can be expected due to increased movement of particles, possibly resulting in the occurrence of turbulence at lower Reynolds numbers than before. Thus, a lower temperature may result in turbulence at a higher Reynolds number. The results of such a study could be used to make the flow more effective in ducts, and consequently, the ideal temperature for ducts used in our society can be found.

Another possible study, based on the same experimental setup, is to use two cameras instead of one since images could be taken from two different angles. In this way, the out-of-plane motion of the particles can be studied as well, thus resulting in a three-dimensional velocity profile illustrating movement of the particles through the whole duct and not only in one plane. This could lead to a better understanding of the movement of the particles and consequently, a better understanding of the behavior of the flow.

Acknowledgement

I would like to thank my mentor’s PHD-student Sagar Zade, who with great kindness, support and patience guided me through this project. I would also like to thank my
mentor Luca Brandth, for making it possible for me to do this project. Furthermore, I owe great thanks to Anna Broms, Serhat Aktay, Klara Kiselman, Isak Nilsson and the entirety of the Rays program who have given help and support when needed, and finally, I want to thank Tekniska Museet and Teknikföretagen, who makes Rays possible in the first place.
References


Appendix

Main Program for PIV Data Processing

1 warning off;
2 cd(pwd);
3
4 % define the parameters ...'
5 setParameters
6
7 for k=1:length(y) % Number of depth planes.
8 Y=ones(size(X))*y(k); % Y co-ordinate in the 'k'th plane.
9 posi=POSI(k);
10
11 % correct position of the measuring points for each camera according to
12 % results of self calibration
13 disp('correcting measuring points locations ...') % Not needed for planar PIV.
14 if selfCalib == 1
15 disp('correction of misalignment...')
16 load([path,'CC_camera1-2_directCC/',posi,'/dispMap.mat'],...
17 'XX','ZZ','Dx','Dz')
18 coeffx=polyfit(XX(:),Dx(:),1);
19 coeffz=polyfit(XX(:),Dz(:),1);
20 x_adjust=polyval(coeffx,X);
21 z_adjust=polyval(coeffz,X);
22 X1=X-x_adjust/2;Z1=Z-z_adjust/2;
23 X2=X+x_adjust/2;Z2=Z+z_adjust/2;
24
25 instDataName = ['/instData_',posi,'_withSC.mat'];
26 else
27 disp('without correction of misalignment...')
28 X1=X;Z1=Z;
29 X2=X;Z2=Z;
30
instDataName = ['/instData_', posi, '__withoutSC.mat'];

% warp measuring points onto image plane of each camera  
[Ic1,Jc1]=define_PIVpoint(X1,Z1,Y,calibData,XPixel_center,ZPixel_center); % Convert mm-point's location to corresponding pixel point's location. These are the points where we want to find the velocity.

if check_projection == 1
  f1=fopen([path,'raw/camera1/',posi,'/Img00020001.bin']);
  bin1 = fread(f1);
  fclose(f1);
  m1=uint8(double(reshape(bin1(5:end),imgSizeJ,imgSizeI)));
  pause();
end

pre-processing of images--------------------------------------------------

disp('pre-processing of images ...')
if exist([path,'raw/camera1/',posi,'/BGB.mat'], 'file') == 0 % i.e. run this loop only if  
  backgroundBlightness(path,posi,imgSizeI,imgSizeJ);
  disp('done')
else
  disp('skipped')
end

PIV calculation ========================================================

disp('PIV processing starts...')
% camera 1 ----------------------------
if isempty(N1) ~= 1
  cam_id=1;
  for i = 1:length(N1)
    tic
    PIV_process_predef_nakaN1_kawata(imgSizeI,imgSizeJ,path,...
    cam_id,posi,Ic1,Jc1,N1(i),lim_ui(1,:),lim_uj(1,:),dfld,ORcoeff,PIVparams,calibData,dt,XPixel_center,ZPixel_center,viewing);
    toc
  end
end
else
disp('PIV proecssing for images by camera0 is skipped ...')
end
end
disp('Program end')

Definition of Parameters used in Main

% In this program, the parameters necessary for PIV algorithm to be run is % defined.

% Experiment condition ==============================================================
Uw = 23.3; %%% wall velocity [mm/s] %%%
dt = 1/117; %%% time interval between image acquisition [s] %%%
hw = 18.5; %%% half width of channel [mm]
temperature = 18.4;

% Digital camera specification ==============================================================
imgSizeI = 768; %%% numberof image elements of CCD (in vertical = x) [pixel] %%%
imgSizeJ = 1024; %%% numberof image elements of CCD (in horizontal = z) [pixel] %%%

N1=1:500; % Camera 1 (Number of Image pairs)
N2=1:2; % Camera 2
num3=1:50; % Stereo.

dx = abs((x(end) - x(1))/(length(x)-1));

% define points at which velocities are measured ==================================
etelemx = 35; % Number of points in x-direction.
x = flipud(linspace(-26,26,etelmx)'); % mm co-ordinates. Flipped because the x-direction
dx = abs((x(end) - x(1))/(length(x)-1));
etelmz = round((etelmx/imgSizeI)*imgSizeJ); % Number of points in z-direction.
z = linspace(-dx*(etelmz-1)/2,dx*(etelmz-1)/2,etelmz);
y=0; % Position of the depth plane.
POSI={'position0'};

viewing =0; % If velocity vectors should be viewed: 1, if not: 0.
check_projection = 1;

[Z,X]=meshgrid(z,x); % These are the grid points where we want to measure the velocity.

% parameters for PIV process===============================================
selfCalib = 0;
path = '../';

% For calibration =========================================================
calibData = '../../calib/PFcoeffs.mat'; % Complex calibration for stereoPIV.

% Specify what is the calibration from the calibration image.
AllconvData = Calibration(path);
calibData = AllconvData(1,1); XPixel_center = AllconvData(1,2); ZPixel_center = imgSizeJ/2;

PIVparams = [48 64 32]; % [largeIA SW smallIA]
lim_ui = [-10 10; -10 10]; % 'lim_ui' and 'lim_uj' define a range of ui and uj (in pixels)
limit_uj = [-10 55; -10 55]; % for outlier removal. The samples which fall outside this range
% recognized as outliers
ORcoeff = [1.6 1.4 1.4 1.2 1.0 0.8]; % coefficients for removing outliers. The first four are
% used in the basic and DWS PIV, and the last two are
% used in the CWS PIV.
% They should be around 1.0. Adjust till the % of outliers is less than 2%.
dfld='dataIA32';

% Kinematic viscosity and Reynolds number  
load('waterProperties.mat')
nu=interp1(Temp,Nu,temperature)*1e-6; % kinematic viscosity (mm^2/sec)

22
Velocity Profile in one Plane

```matlab
Re=Uw*hw/2/interp1(Temp,Nu,temperature);

cd('/home/username/Documents/Takuya/PlanarPIV');

imageIdx=1:500; % Number of image pairs.
dfld='dataIA32';

% POSI = ['position0';'position1';'position2';'position3';...
% 'position4';'position5';'position6';'position7';'position8'];
POSI = 'position0';

for i=1:size(POSI,1)
    posi=POSI(i,:);

    path1=['../',dfld,'/camera1/',posi];

    Ui = 0; Uj= 0;
    for k=1:length(imageIdx)
        if imageIdx(k) < 10
            fid = ['000', int2str(imageIdx(k))];
        elseif imageIdx(k) < 100
            fid = ['00', int2str(imageIdx(k))];
        elseif imageIdx(k) < 1000
            fid = ['0', int2str(imageIdx(k))];
        else
            fid = int2str(imageIdx(k));
        end
        load([path1,'/data',fid,'.mat']) % ['CCtemp', 'uj', 'ui', 'xj','xi','erruj','errui','calibData','dt']
        Ui = ui + Ui; Uj = uj + Uj;
    end
   %%%%%%
    Umeani(:,:,i)=(Ui./length(imageIdx)).*(calibData/dt);
    Umeanj(:,:,i)=(Uj./length(imageIdx)).*(calibData/dt);
```
\( \text{Xi}(i,j) = (x_i - \text{XPixel}_\text{center}).*(\text{calibData}); \)

\( \text{Xj}(i,j) = (x_j - \text{ZPixel}_\text{center}).*(\text{calibData}); \)

\% Mean velocity profile.

\( \text{Umeanprofi}(i) = \text{mean}(	ext{Umeani}(i,j),2); \)

\( \text{Umeanprofj}(i) = \text{mean}(	ext{Umeanj}(i,j),2); \)

end

save([path1,'/Postprocc/','Mean.mat'],'Umeani','Umeanj','Xi','Xj');

figure(2)

plot(Umeanprofj,Xi(:,1),'o-'); hold on;
plot(Umeanprofj,zeros(size(Umeanprofj)));
hold off;
xlabel('Velocity: mm/s'); ylabel('x: mm');

\textbf{Three-Dimensional Velocity Profile}

cd('/home/username/Documents/Takuya/PlanarPIV');

imageIdx=1:50; \% Put the right number of image pairs.

dfld='dataIA32';

POSI = ['-20mm';'-15mm';'-10mm';...
' -05mm';'+00mm';'+05mm';'+10mm';'+15mm';'+20mm'];

\% POSI = 'position0';

for i=2:size(POSI,1)+1

posi=POSI(i-1,:);

path1=['../',dfld,'/camera1/','position0/',posi];

Ui = 0; Uj= 0;

for k=1:length(imageIdx)
if imageIdx(k) < 10
    fid = ['000', int2str(imageIdx(k))];
elseif imageIdx(k) < 100
    fid = ['00', int2str(imageIdx(k))];
elseif imageIdx(k) < 1000
    fid = ['0', int2str(imageIdx(k))];
else
    fid = int2str(imageIdx(k));
end
load(['path1','/data',fid,'.mat']) % ['CCtemp', 'uj', 'ui', 'xj','xi','erruj','errui','calibData','dt']
Ui = ui + Ui; Uj = uj + Uj;

Umeani(:,:,i)=(Ui./length(imageIdx)).*(calibData/dt);
Umeanj(:,:,i)=(Uj./length(imageIdx)).*(calibData/dt);
Xi(:,:,i) = (xi - XPixel_center).*(-calibData);
Xj(:,:,i) = (xj - ZPixel_center).*(calibData);

% Mean velocity profile.
Umeanprofi(:,i) = mean(Umeani(:,:,i),2);
Umeanprofj(:,i) = mean(Umeanj(:,:,i),2);
Ximean(:,i) = mean(Xi(:,:,i),2);
end

Umeanprofi(:,1) = 0; Umeanprofi(:,size(POSI,1)+2) = 0; % For plane -25 and 25.
Umeanprofj(:,1) = 0; Umeanprofj(:,size(POSI,1)+2) = 0; % For plane -25 and 25.
Ximean(:,1) = Ximean(:,2); Ximean(:,size(POSI,1)+2) = Ximean(:,size(POSI,1)+1); % For plane -25 and 25.

Z = [-25 -20 -15 -10 -5 0 5 10 15 20 25];
Zimean = ones(size(Ximean,1),1)*Z;
surf(Zimean,Ximean,Umeanprofj);
xlabel('Z'); ylabel('X');
commandwindow

disp('Welcome.........So do you want to calculate the velocity profile in a rectangle?');
disp('If yes then, press enter');
pause()
a = input('Good. What is the width of the rectangle in mm?');
b = input('Thanks. Now, what is the height of the rectangle in mm?');
beta = b/a;

disp('Please check if the rectangle is drawn correctly...... and if it is OK, press enter');
figure(1);
pos = [0 0 a b];
rectangle('Position',pos,'FaceColor',[0 .5 .5],'EdgeColor','b','LineWidth',2);
xlim([-0.5*a,1.5*a]);
axis equal
pause()

Ndiv = input('What is the resolution that you want: Coarse = 10 or Fine = 100......?');
countx = 0;
for x = linspace(0,1,Ndiv)
countx = countx + 1;
county = 0;
for y = linspace(0,beta,Ndiv)
county = county + 1;
Vtemp = 0;
for n = 1:2:10
    for m = 1:2:10
        Vtemp = Vtemp + (16*(beta^2)/(pi^4))*sin(n*pi*x)*sin(m*pi*y/beta)/(n*m*((beta^2)*n^2 + m^2));
    end
end
V(countx,county) = Vtemp;
end
end

26
figure(1);
contourf(a.*linspace(0,1,Ndiv),a*linspace(0,beta,Ndiv),V);
axis equal
pause()

commandwindow

Cond = input('Do you want to see some more amazing plots? If yes, press 1. If you are happy, press 0');

if Cond == 1
    figure(2);
    plot(2)%subplot(2,2,1)
    surfc(a.*linspace(0,1,Ndiv),a*linspace(0,beta,Ndiv),V,V,'LineStyle','none');
elseif Cond == 0
    return
else
    end